Pro/ENGINEER® 2001

Pro/DETAIL™
Topic Collection

Parametric Technology Corporation

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Pro/DETAIL

To Create Witness Lines

1. Choose **ATTACH TYPE** > **Make Line**.
2. Using the **MAKELINE** menu, do one of the following:
   – Choose **2 Points** and select two vertices. The system highlights them in white and draws a blue phantom font line to indicate the line that you constructed between them.
   – Choose **Horiz Line** or **Vert Line** and select a vertex through which the horizontal or vertical witness line is to pass. You can select a datum curve endpoint, solid vertex, coordinate system origin, datum point, cosmetic feature endpoint, or draft vertex. The system highlights the vertex in white, and displays two blue arrows pointing horizontally or vertically away from the vertex.

About Modifying Witness Lines

When Pro/ENGINEER places a dimension on a model, it leaves a gap between the model and the witness line. The drawing setup file option `witness_line_offset` controls the actual size of the gap. Sometimes this gap is not visible on a drawing. However, it becomes noticeable when you plot the drawing. To see how the drawing will look when you print it, choose **File > Print > Screen**.

A witness line can have both jogs and breaks. When you add jogs to a witness line that already has breaks, the breaks that were created as simple breaks appear on the first unjogged segment of the witness line. If a witness line has dimension breaks, jogging a witness line relocates a break to a new intersection point.

You can edit witness lines in the following ways:

- Create parametric or simple breaks in dimension witness lines and in leader lines of notes and symbols.
- Resize the gap by using the **Move** command. You can simultaneously clip several dimensions that have the same orientation so that their witness line endpoints all align. Using multiple selection, you can manipulate witness lines in the following ways:
  – Clip both witness lines of a dimension at the same time.
  – Clip the witness lines of many dimensions at once. When you pick on the witness line of each dimension, the system moves each of the endpoints together.
- Erase and resume witness lines.
- Skew witness lines.
- Add jogs to and delete them from linear (standard and ordinate) and angular dimension witness lines, note leaders, and symbol leaders.
- Create and modify angular, diametrical, radial, and linear dimensions with and without an elbow.

To Clip a Witness Line

1. Select the dimension with the witness line that you want to clip. Use the middle mouse button to end the selection.
2. On the right mouse button pop up menu, select **Move/Activate**. The witness lines are highlighted, with a "handle" at each end.
3. Drag the handles to the desired positions. Left click to finish.

To Erase a Witness Line

You can erase only one witness line of a dimension. If you attempt to erase the second one after erasing the first one, the system erases the second one and restores the first one. You cannot erase both.

1. Select the dimension with the witness line to erase.
2. Click the right mouse button, and select **Erase Wit Line** from the pop up menu.
3. Click the witness line to erase. The system erases the dimensions and then redraws them. It blanks the witness line and displays a double arrowhead in its place.

Note: You cannot erase the witness lines of ordinate dimensions (including ISO-standard ordinate dimensions

with a cross-line between the baseline and the dimension).

You cannot blank the witness line of a clipped dimension (one that has been clipped as a result of view clipping), and you cannot use **Show** to restore a witness line that has been automatically erased (clipped due to view clipping).

To Restore an Erased Witness Line

1. Select the dimension with the witness line to restore.
2. Click the right mouse button, and select **Show Wit Line** from the pop up menu. The system redraws the selected dimension and displays its witness line with a single arrowhead at the leader.

To Create a Break in a Leader or Witness Line

1. Select the dimension with the witness line to break. The dimension is highlighted.
2. From the right mouse button pop up menu, click **Break**.
3. On a line, select the first point of the break.
4. Select a second point on the same line to define the extent of the break. The system creates a gap between these two points.
5. Use the line "handles" to size the break.

Note: You cannot break a radial or diameter dimension of an arc or circle because what you see is not a witness line, but the extension of a dimension arrow.

To Create a Parametric Break at an Intersection of Witness Lines

You can make parametric breaks against dimension witness lines or snap lines. When you create a break against a snap line, the break moves when the snap line moves, and is deleted when the other dimension is erased or deleted.

Note: Parametric breaks cannot be created against note or symbol leaders.
1. Select the dimension with the witness line to break. The dimension is highlighted.
2. From the right mouse button pop up menu, click **Break**.
3. On a line, select the first point of the break.
   The BREAK TYPE menu appears in the Menu Manager, letting you create either a simple break or a parametric break. Selecting from this menu is optional—the system assumes that the break is a parametric break if your next pick is on the witness line of another dimension or on a snap line. Both **Simple** and **Parametric** are selected by default. Clear the **Simple** check box to ensure that a parametric break is created.
4. Select a witness line of another dimension or a snap line against which to break.
5. Using the BREAK SIZE menu in the Menu Manager, choose one of the following commands:
   – Choose **Default** and specify the gap size for the selected intersection point. The system assigns the default break size, which is equal to twice the `witness_line_offset` distance specified in the drawing setup file.
   – Choose **Pick** and set the break gap size by dragging the pointer along the witness line and using the left mouse button to finish selecting gap size.
   – Choose **Radius** and specify a radius value for the break.
     The system creates a parametric break at the closest intersection point of the originally selected witness line and the second witness line or snap line.
6. To delete *all of the breaks* pertaining to the selected dimension, choose **BREAK** > **Remove**.

To Modify Witness Lines

1. Select the dimension with the witness line to edit. The dimension is highlighted.
2. From the right mouse button pop up menu, click the action you want to perform.

You choices are:

Move\Activate - Move label and arrows
Default Wit Line - Toggle the default witness line style.
Switch View - Move a selected dimension object from one view to another.
Make jog - Insert a corner in a line.
Nominal Val - Change selected dimension value.
Values/text - Open a dialog box to edit values and appearance of dimension.
Flip Arrows - Toggle arrows inside or outside witness lines.
Arrow Style - Change head style of selected arrows.
Break - Insert a break in a witness line.
Del Breaks - Remove previously made breaks.
Erase Wit Line - Hide a selected witness line.
Erase - Remove the dimension

To Skew Witness Lines

1. Select the dimension with the witness line to edit. The dimension is highlighted.
2. Use the cursor to select a handle on the end of a witness line. The cursor changes to an arrow, perpendicular to the selected witness line.
3. Drag the handle to the desired location. Left click again to place the witness line.
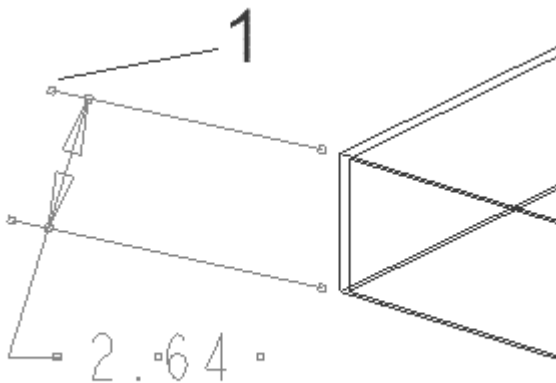
Drag handle (1) up and down to skew dimension position.

To Add a Jog to a Witness Line

1. Choose **Insert** > **Jog**.
2. Select the dimension to which you are adding a jog.
3. Select a point on a witness line. The remaining portion of the witness line moves with the pointer.
4. Select locations for jog points by pressing the left mouse button.
5. Press the middle mouse button. The system locates the jog.
   Notes:
   The first and last segments of a jogged witness line are always parallel.
   When the system locates the jog, it remains in the same place on the drawing, regardless of where you move the dimension.

About Using Dual Dimensioning

Using the drawing setup file option `dual_dimensioning`, you can display dimension text in two sets of units—primary or secondary (or both). These drawing setup file options work together with `dual_dimensioning` to modify the display of dual dimensions:

- `dual_secondary_units` specifies the secondary set of units used by the drawing.
- `dual_digits_diff` specifies the number of decimal places that secondary units contain compared with the primary units used.
- `decimal_marker` specifies the character to use to mark the decimal point in secondary dimensions.
- `dual_dimension_brackets` specifies whether one of the units of dimensions appears within brackets.

    Note: When you are using dual dimensioning with dimensional tolerances, the system rounds the secondary values so that they always fit within the range indicated by primary units (that is, so that they are *tighter* than primary values), and retain the desired design intent.

To Place Dual Dimensions in a Drawing

To place dual dimensions, you can use various options in the Modify Dimension dialog box:

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Modify Dimension** dialog box opens.
- To place the secondary dimension below the primary dimension, select **Below** from the **Dual Dimensioning** field
- To place the secondary dimension on the same line as the primary dimension, select **To Right** from the **Dual Dimensioning** field.
- To specify the number of decimal places to use for the secondary dimension, type a value in the **Dual Number of Digits** text box
3. Click OK.

Note: If you are using metric dimensions as primary or secondary units in dual dimensions, the metric dimensions do not display as fractions if the fraction format is set.

To Add Text to a Dimension

Use this procedure to add prefix or "postfix" text to a dimension.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Modify Dimension** dialog box opens.
3. Click the **Dim Text** tab.
4. Use the **Name**, **Prefix** and **Postfix** boxes to surround the dimension text.
5. Click **OK**. The system stores the dimension text with the model, and displays it in either Part mode or Assembly mode when you select the feature the dimension is driving.

To Add Parameters to Dimension Text

Use this procedure to add prefix or "postfix" text to a dimension.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Modify Dimension** dialog box opens.
3. Click the **Dim Text** tab.
4. Type the parameter callout in the **Prefix** or **Postfix** field.
5. Click **OK**.

Note: You can place dimensions and parameter text in driven dimensions or associative dimensions owned by the same model; however, you cannot place them in draft dimensions or associative draft dimensions.

To Overwrite Dim Value Display with a Text String

Use this procedure to replace any dimension value display with your own text string.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Modify Dimension** dialog box opens.
3. Click the **Dim Text** tab.

4. In the text field, replace the symbol "@D" with "@O"; then type the desired text. The symbol "O" represents *overwrite*; anything you type following this text overwrites the dimension value. The drawing now displays the text string instead of the dimension value.

Note: Dimensions must always have a displayed value. If you do not type text after substituting "@O," the system ignores the procedure when modifying driven dimensions and displays the "REF" text when modifying reference dimensions.

To Modify Dimension Decimal Places

You can modify the number of decimal places a dimension displays individually or for a multiple selection of dimensions.

For an individual dimension:

1. Select the dimension
2. From the right mouse button pop up menu, click **Values / Text**.
3. In the Number of Digits field, enter the number of decimal places.
4. Click Ok.

For multiple selection:

5. Click **Format** > **Number of Digits**.
6. In the prompt line, enter the value you want to assign.
7. Use the **Get Select** > **Pick Many** command on the **Menu Manger** toolbar to draw a selection rectangle around all the dimensions you want to edit.
8. Click **Done Sel**. The decimal places are changed.

Notes:

To control the display of the dimension value for primary and secondary units in the drawing *independently* (whether the dimension value has leading or trailing zeros), use the drawing setup file option `lead_trail_zeros`.

To specify whether the character used to indicate the decimal point in a dimension is a period, a comma, or a comma only when the units are metric, use the drawing setup file option `decimal_marker`.

To Show Dimensions in Degrees or Minutes/Seconds

For an individual dimension:

1. Select the dimension (Hold down the Shift key for multiple selections.)
2. From the right mouse button pop up menu, click **Values / Text**.
3. In the **Format** field, click **Fractional**.
4. Click **Ok**.

For minutes/seconds:

Type the value [3600] in the **Largest Denominator** box. You can type a value of 60 if the drawing setup file option `draw_ang_units` is set to `ang_min` since this value essentially reduces any dimension to an integer with minutes precision.

Setting the Default Decimal Places and Trailing Zeros

• Change the number of decimal digits shown by setting the configuration file option `default_ang_dec_places` to a value of `0` through `14` (the default is `1`).

• Remove the trailing zeros to conform to ANSI standards by setting the drawing setup file option `draw_ang_unit_trail_zeros` to `yes` (the default). This applies *only* if the entire seconds or

seconds/minutes field is exactly zero. Select **Fractional** and type [3600] in the **Largest Denominator** box to be sure.

To Display Dimension Text Symbols

1. Select the dimension (Hold down the **Shift** key for multiple selections.)
2. From the right mouse button pop up menu, click **Values / Text**.
3. In the **Modify Dimension** dialog box, click **Dim Text**.
4. In the **Dimension Text** box, note the current @D setting. Backspace over the D and replace it with an S.
5. Click **OK**. The selected dimensions now display their symbolic values rather than their dimension values. Type a prefix or a postfix if you want to add text to the value of the dimension.

To toggle the display of all dimensions temporarily, choose **Info > Switch Dims**

To Modify the Value of Dimension Symbols

1. Select the dimension (Hold down the Shift key for multiple selections.)
2. From the right mouse button pop up menu, click **Values / Text**.
3. In the **Modify Dimension** dialog box, the value in the **Name** box is the current dimension symbol. Type new text to change the symbol value.
4. Click **OK**. The system stores the new dimension symbol as modified in the model and updates any relations using the symbol.

To Show an Angular Dimension in Degrees, Minutes and Seconds

1. Select the dimension (Hold down the Shift key for multiple selections.)
2. From the right mouse button pop up menu, click **Values / Text**. The **Modify Dimension** dialog box opens.
3. From the **Angular Dim Units** field, select one of the following from the drop-down list:
   – **Degrees**—Shows the angular dimension in degrees
   – **Degrees, Min.**—Shows the angular dimension in degrees and minutes
   – **Degrees, Min., Sec.**—Shows the angular dimension in degrees, minutes and seconds.
4. The angular dimension is changed to the specified unit.

Note: You can also change the way you show angular dimensions globally by using the drawing setup file option `draw_ang_units`.

About Cleaning Up Dimensions

Using the **Edit** > **Clean Dimensions** command, you can arrange the display of selected dimensions automatically by doing any or all of the following:

- Clean dimensions by view, or by individual dimensions.
- Center dimensions between witness lines (including the entire text box with gtols, diameter symbols, tolerances, and so on).
- Create breaks in witness lines where they intersect other witness lines or draft entities.
- Place all dimensions on one side of a model edge, datum plane, view edge, axis, or snap line.
- Flip arrows.
- Offset dimensions from edges or view boundaries.

To Clean Dimensions

Use this procedure to automatically clean up multiple dimensions at once.

1. Choose **Edit > Clean Dimensions**. The **Clean Dimensions** dialog box opens.
2. Using the **Get Select** options on the Menu Manager, select individual dimensions or an entire view; then

click **Done Sel**. The **Clean Dimensions** dialog box activates.

Placement Tab

3. Type an initial uniform offset value in the **Offset** box. To place the selected dimensions, type an incremental offset value in the **Increment** box.
4. In the **Placement** page, do one of the following:
   – Select **View Outline** to offset the dimensions with respect to their view outlines. *Go to Step 6.*
   – Select **Baseline** to reposition only dimensions in the same view whose leaders are parallel to the selected baseline. *Go to Step 5.*
5. Select one of the entities in the drop down list to use as a cleaning baseline. An arrow appears to indicate the offset direction. To change the direction click **Flip Arrow**.
6. Check **Create Snap Lines** to add dashed snap lines to the dimensions.
7. Check **Break Witness Lines** to break witness lines where they intersect other draft entities.
8. Click **Apply**. The system applies cosmetic cleanups to all dimensions. Click **Undo** to return them to their pre-cleaned state and try again without reselecting dimensions.

Cosmetic Tab

9. In the **Cosmetic**.tab, by default, the system selects all of the check boxes. Clear any or all of them, as necessary; then click **Apply**:
   – **Auto flip arrows** flips the arrows inside the witness lines if they fit (without overlapping the text); if they do not fit or they would be overlapping the text, they flip outside the witness lines.
   – **Center text** centers the text of each dimension between its witness lines. If it does not fit, the system moves the text outside the witness lines in the specified direction:
     Horizontal moves the text to the left or right.
     Vertical *moves text of vertical dimensions up or down.*
   – **Create snaplines** creates snap lines under all moved dimensions (if the destination does not already have a snap line in the vicinity). They appear only under dimensions that are parallel to baselines (if you have selected a baseline) or parallel to a view border (if the view outline has been selected). Once you clear this check box, it remains cleared for the remainder of your Pro/ENGINEER session.

To Relate Detail Items and Dimension Text

You can directly relate a note, draft geometric tolerance, or symbol to dimension text so that the detail item moves with the dimension when the dimension changes location.

To relate an existing detail item to dimension text,

1. Click **TOOLS** > **Relate Obj** > **Add Items**.
2. Select the "host" object.
3. Select all other items you want to relate to the host.
4. Click **Done Sel**. All the objects are grouped and selected as one.

To remove objects from the group, use **TOOLS** > **Relate Obj > Remove Items**.

Note: When creating a *new* symbol, note, or free draft geometric tolerance, you can relate it to a dimension when specifying its placement.

To Set Default Dimension Text Orientation

You can control the positioning of dimension text relative to the leader lines by setting the drawing setup file option `text_orientation`.

- If you set it to `horizontal` (the default), the dimension text always appears horizontally, centered with regard to the leaders.
- If you set it to `parallel`, the dimension text always appears parallel to the leader lines, regardless of their orientation. You can use the drawing setup file option `parallel_dim_placement` to specify whether the dimension values appear above or below the leaders. The dimension text must be parallel in order for this option to have effect; it does not have any effect when dual dimensions are present.

The drawing setup file option `angdim_text_orientation`—not `text_orientation`—controls the display of angular dimensions.

As illustrated in the following figure, `angdim_text_orientation` has the following values: `horizontal`, `horizontal_outside`, `parallel_outside`, `parallel_above`, and `parallel_fully_outside`.

Displaying Text of Angular Dimensions
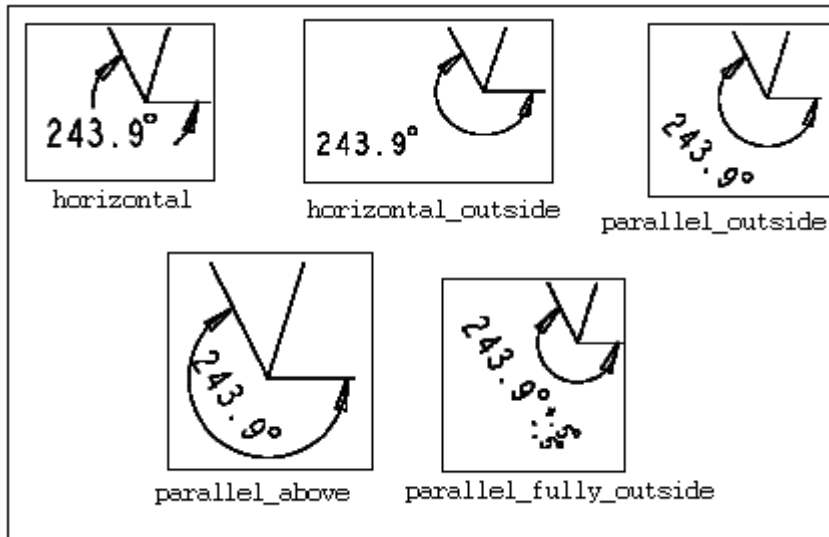
When you retrieve a drawing that was created before Release 15.0, the values translate as follows:

- If the value of `text_orientation` is `parallel`, the value of `angdim_text_orientation` is `parallel_outside`.
- If the value of `text_orientation` is `horizontal`, the value of `angdim_text_orientation` is `horizontal`.

To Set The Default Chamfer Dimension Text Display

By setting the configuration file option `chamfer_45deg_dim_text`, you can control the display of chamfer dimension text without affecting the leader. This only affects the text of *newly created* dimensions. To modify dimensions that you created before changing the setting, you must manually edit the text. Using the drawing setup file option `chamfer_45deg_leader_style`, you can control the leader type of a chamfer dimension without affecting the text. You must use `chamfer_45deg_dim_text` and `chamfer_45deg_leader_style` together to meet the appropriate standard. To control the use of leading or trailing zeros in the dimension value, use the drawing setup file option `lead_trail_zeros`.

About Dimensional Tolerances in Pro/DETAIL

You can set the tolerance standard as ANSI or ISO, and set the tolerance display on or off. You can drive dimensional tolerances using a set of tolerance tables. The system assigns each model a tolerance standard of either ANSI or ISO.

- When you switch from ISO to ANSI, the system assigns the ANSI tolerances based on the nominal dimension's number of digits and deletes the tolerance table reference.
- When you switch from ANSI to ISO, a set of tolerance tables drives the ISO-standard tolerances.

To Create an ISO-standard Model in Drawing Mode

1. Click **DRAWING > ADV DWG OPT > TOL SETUP > Standard > TOL STANDARD > ISO/DIN. or ANSI**
Pro/ENGINEER loads the system and user-supplied tables, and the General table drives all dimensions.

Note:

The system loads the tolerance tables into the model when you create it as an ISO-standard model or switch the tolerance standard from ANSI to ISO. To create the model as an ISO-standard model, set the configuration file option `tolerance_standard` to `ISO`. Since the tables determine how the model regenerates, the system stores them permanently with the model, and you can only use them with driving dimensions. Four types of tolerance tables are available:

- General (one per model)
- Broken Edge (one per model)
- Holes (several per model)
- Shafts (several per model)

When you create a dimension, the system assigns it to the General table. When you assign the dimension to the tolerance table, the tolerance table and its dimension value now govern the tolerance values of the dimension. You can switch the tolerance table reference of the dimension to any other table. Dimensions in ISO models, which are driven by Holes or Shafts tables, appear as follows:

```
PLUS MINUS—5.69 f7 (+0.01)
NOMINAL—30 f7 (5.78)
LIMITS—5.69 f7
```

To Change the Tolerance Class

Each ISO-standard model has an extra attribute called the *tolerance class* which determines the general coarseness of the model. The configuration file option `tolerance_class` sets the default tolerance class for ISO models (the default is `medium`). The system uses the tolerance class together with the dimension value when retrieving tolerances for General or Broken Edge dimensions.

1. Click **DRAWING > ADV DWG OPT > TOL SETUP** > **Model Class**.
2. From the **TOL CLASSES** menu, select a class name.
3. All dimensions driven by the General or Broken Edge table obtain new tolerance values. Regenerate the model.

Loading the System and User-Supplied Tables

The configuration file option `tolerance_table_dir` sets the default directory for a user-defined tolerance table. All Holes and Shafts tables overwrite existing tables when loaded. When loading General and Broken Edge tables, keep in mind the following:

- If you load one table that has the same set of class names as the model's, the system accepts the new table.
- If you load a table that contains class names that conflict with those already loaded in the system, the system does not load those class names.
- If you load two tables with class names that do not conflict with those in the system, but that are different from them, you overwrite the ones in the system.
- If the default class of the model does not exist in the new names, you must specify a new class.

To Load System and User-Supplied Tables

After you load the new tables, the system assigns the new dimension tolerances and you can regenerate the model.

When you regenerate the model by choosing **Regenerate** from the DRAWING menu and **Model** from the

REGENERATE menu, the system goes through all of the dimensions and reassigns their tolerances from the tolerance tables. If you modify a dimension tolerance, the system deletes the tolerance table reference for that dimension, and the tolerance value remains the same until you modify it again or reassign the tolerance table.

If the regeneration fails, all relevant dimensions acquire the backup tolerance values. However, the new tolerance tables remain.

Example: A Tolerance Table

TABLE_TYPE	HOLES				
TABLE_NAME	A				
TABLE_UNIT	MICROMETER				
RANGE_UNIT	MILLIMETER				
BASIC SIZE	9	10	11	12	13
—3	295/270	310/270	330/270	370/270	410/270
3 - 6	300/270	318/270	345/270	390/270	450/270
6 - 10	316/280	338/280	370/280	430/280	500/280
10 - 18	333/290	360/290	400/290	470/290	630/300
18 - 30	352/300	384/300	430/300	510/300	700/310
30 - 40	372/310	410/310	470/310	560/310	710/320
40 - 50	382/320	420/320	480/320	570/320	800/340
50 - 65	414/340	460/340	530/340	640/340	820/340

Changing the Tolerance Table Reference

All instances in a family share the same set of tolerance tables, the same tolerance standard, and the same class. When changing the tolerance table reference, keep in mind the following:

- If you modify model units, but keep all of the dimension values the same, the system updates the tolerance values to reflect the change in the overall model size.
- If a Holes or Shafts tolerance table drives a dimension's tolerances, you cannot show it in a plus-minus symmetric format. The system assumes that the General and Broken Edge tables have symmetric values.
- If you place a dimension tolerance in a family table, the system deletes its tolerance table reference. Also, if you switch a model from ANSI to ISO, or vice versa, it preserves the tolerances in the family tables and does not assign table references to those dimensions.
- If a dimension value falls outside ranges specified in the table, the system uses the closest range to obtain tolerances (that is, it uses the last range in the system table (2000–4000) to determine tolerances for dimension values of 2000 or greater).

To Change the Tolerance Table Reference

1. Choose TOL SETUP > **Tol Tables**. The TOL TBL ACT menu displays these commands:
   - **Modify Value**—Displays the tables in the TOL TABLES menu. You can modify their contents using Pro/TABLE.
   - **Retrieve**—Retrieves a set of tables into the model.
   - **Save**—Saves the tolerance table.
   - **Show**—Displays the tolerance table, as shown next.

TABLE_TYPE	GENERAL
TABLE_NAME	DEFAULTS
TABLE_UNIT	MILLIMETER
RANGE_UNIT	MILLIMETER

DESCRIPTION	0.05–3
FINE	0.05–
MEDIUM	0.1
COARSE	0.2
VERY COARSE	0.5

2. Do one of the following:
    – Choose **Modify Value**, select the table by choosing **General Dims** or **Broken Edges**, and select the dimensions.
    – Choose **Holes** or **Shafts**; then type the table name and class number

Setting the Tolerance Display

To set the tolerance display on and off, use the drawing setup file option `tol_display`. You can set the default display for dimension tolerances using the configuration file option `tol_mode`.

Using the configuration file option `maintain_limit_tol_nominal`, you can maintain the nominal value of a dimension regardless of the changes that you make to the tolerance values. If you set it to `yes`, the system does not modify the **Nominal Value** of a dimension with a **Limits** tolerance format when you set the format to **Limits** or change the value of the upper or lower tolerance.

To Set the Tolerance Display for Individual Dimensions

1. Select a dimension.

2. From the right mouse button pop up menu, click **Properties**.

3. In the Modify Dimension dialog box, select a tolerance format from the **Tolerance Mode** list. Specify values in the appropriate fields:
    – For the **Limits** format, specify values in the **Upper Tolerance** and **Lower Tolerance** boxes.
    – For the **Nominal** and **Plus-Minus** formats, specify values in the **Nominal Value**, **Upper Tolerance**, and **Lower Tolerance** boxes.
    – For the **+-Symmetric** and **As Is** formats, specify a value in the **Nominal Value** box.
4. Click **OK**. When tolerances appear, the system lists the default tolerance values in the lower-right corner of the screen. .

To Modify Dimensional Tolerances in a Note

You can create a note in which you substitute a dimensional tolerance, entered as a parameter, by its value. The following figure illustrates the format to use. You can use a similar format to include an angular tolerance value in a note. For example, type [&angular_tol_0_0] to display "0.5" (the default for one decimal). [&angular_tol_0_00] to display 0.05 etc.

Because the system includes tolerances in a note as parameters, associativity exists between the tolerance value in a note and a model. When you change tolerance values in a note, the system updates the default tolerance table in Part mode.

To modify the tolerance value in a note,

1. Select the note.
2. From the right mouse button pop up menu, click **Edit Text.**
3. Type a new value for the tolerance. Click OK.
Note: Tolerance tables do not affect thread dimensions. Thread dimensions come into the model having the same tolerances with which the system stored them.

About Drawing Notes

In Drawing mode, a note can be part of a dimension, attached to one or more edges on the model, or "free." Pro/ENGINEER creates note text using the default values (such as height, font) specified in the **Format > Text Style Gallery** dialog box commands.

During the notemaking process, you specify the following characteristics:

- Leaders
    - No leader or multiple leaders
    - A leader attached to a model edge or draft entity, pointing anywhere on the drawing
    - A leader attached to a model edge or datum point
    - An ISO-standard leader line
    - A leader that is normal to an entity
    - A leader that is tangent to an entity
- Format/Placement
    - Horizontal
    - Vertical
    - Displayed at a specified angle
    - Left-justified
    - Right-justified
    - Centered
    - Related to dimension text
- Text style

After you create your first note, Pro/ENGINEER creates subsequent notes using those attributes that you specified previously.

The ATTACH TYPE Menu In Pro/DETAIL

- **On Entity**—Creates a note with a leader that is attached to a model edge or draft geometry. Selecting the entity near its vertex locates the leader at the vertex.
- **On Surface**—Locates the leader on model geometry or surfaces. You can select model geometry, threads, and datum surfaces. Once you make an attachment on a surface, if you reorient the view, the attachment remains. If the size of the surface changes, the system updates the leader point accordingly.
- **Free Point**—Locates the leader anywhere on the drawing. You must have a Pro/DETAIL license to create a note using this command.
- **Midpoint**—Locates the leader at the midpoint of a model edge or draft geometry. The next figure illustrates where you can find the midpoint of entities such as splines, arcs, circles, and ellipses.
- **Intersect**—Locates the leader at the intersection of two model edges or two draft entities.
- **Arrow Head**—Creates a leader with an arrow head.
- **Dot**—Creates a leader with a dot.
- **Filled Dot**—Creates a leader with a filled dot.
- **No Arrow**—Creates a leader with no arrow.
- **Slash**— Creates a leader with a slash.
- **Integral**—Creates a leader with an integral.
- **Box**—Creates a leader with a box.
- **Filled Box**—Creates a leader with a filled box.

To Add a Drawing Note

1. Choose **Insert** > **Note**.
   Use the **Note Types** menu on the Menu Manager to specify the note appearance and contents source:
   Note Type:

   > **No Leader**— No leader
   > **Leader**— Leader attached to specified point.
   > **ISO Leader**—Two leaders converging in one note.
   > **On Item**—No leader, note is directly attached to selected item.
   > **Offset**—No Leader, select a point offset from a selected draft item.

   Note Source, (Enter or File) and if a leader is used, leader orientation,

   - **Standard**—Uses the default leader type.

   - **Normal Ldr**—Makes the leader normal to the entity; in this case, the note can have only one leader.

   - **Tangent Ldr**—Makes the leader tangent to the entity; in this case, the note can have only one leader.

2. Still n the same menu, specify the alignment, (Left, Center, Right) and text style.
   (By default the note is made with the current style, or last one you used. Use **Cur Style** to select a new current style from a list of defined styles, or click **Style Lib** to define a new style. or select one from the library.)

3. When you have finished, choose **Make Note**. You are prompted to supply the contents of the note either by selecting a file or by typing in the prompt line. Select the location on an edge or datum point for the note.

4. Type an angle for the note between 0 and 359 degrees.
   Note: You cannot move notes that you created using **On Item**, but you can modify them in all other ways.

To Relate a Note to Dimension Text

1. Choose DETAIL > **Create** > **Note** > **Dim Related** and any of the other available commands from the NOTE TYPES menu.
2. When you have finished, choose **Accept**.
3. Select dimension text and a location relative to the text.

To Enter a Note from a File

1. Choose **Insert** > **Note**. The NOTE TYPES menu opens on the Menu Manager.
2. Select all the norte-descriptive commands you need, including **File**. Click **Done**.
3. Select a location on the drawing for the note.
4. From the SELECT FILE menu, do one of the following:

   - Use the **Enter NAME:** command to retrieve a note from a specific directory and type a filename, complete with its extension (for example, filename.xxx, where xxx can be any combination of alphanumeric characters).

   - Use the **Current Dir** command to retrieve a note from the current directory.

   - Use the **Note Dir** command to retrieve a note from the default note directory. Pro/ENGINEER does not save notes by default with the ".txt" extension. If a filename does not have a ".txt" extension, it does not appear in the default note directory.

   The note appears in the specified location.
   Note: You should not include tabs in the file since they justify the note text improperly on the drawing.

Entering Notes from a File or the Keyboard

When you are entering notes from a text file, the file can reside in the current directory, or can be present anywhere within a search path that you have specified using the configuration file option "pro_note_dir." You can enter notes from a file that contain dimensions, parameters, special symbols, and superscripted or subscripted text. However, you cannot enter information about characteristics such as text height, text width, text angle, and slant angle. You must use the commands in the MODIFY TEXT menu to change this information manually.

When you use the keyboard to type note text manually, you can add blank lines, create superscripted and subscripted text, add symbols, and include parameter information.

To Add a Blank Line to a Note

When typing the text of a note, press the ENTER key on a blank line to exit the note creation.

To include a blank line in the note text, press the SPACE bar and then the ENTER key.

To add a blank line between text strings in a single note, use the **Full Note** command to edit and add the blank lines there.

To Write Notes to a File

1. From the Pro/ENGINEER menu bar, choose **Info** > **Save Note**.
2. Select a note to save.
3. Type the name of a destination file. If necessary, select another note and type the filename.
4. Choose **Done Sel**.

Creating and Saving Drawing Notes

Using the Pro/ENGINEER **Info** menu, you can write the following information to a file:

- Annotations to views.
- Notes with or without a leader, containing any number of lines.
- Parameters (the system substitutes parameters with their values after copying them to a file).

Notes written to disk using **Info** > **Save Note** are automatically appended with a .txt extension and a version increment extension. This causes notes to appear automatically in the menus when you retrieve them when you are creating notes.

The system saves each selected note in a separate file. When specifying the name of a destination file, type the file name without an extension. The system then automatically appends the default .txt extension and a version increment extension (for example, note.txt.1). You can save notes with another extension (for example, note.ascii.1). The system uses your extension instead of .txt and appends a version increment extension (note.ascii.1). If you type the same file name as an existing file name, the system updates the version increment extension automatically so as not to overwrite an existing file (note.txt.2).

Note: During the process, Pro/ENGINEER loses boxes and underlining information, and does not save special symbols. To save them, bring the note into a text editor and save it from within the editor.

Example: Creating a Balloon Note
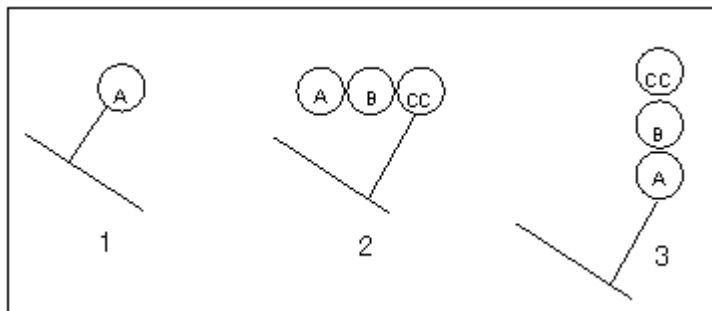
1. A single balloon note.
2. A horizontally stacked balloon note.
3. A vertically stacked balloon note.

To Add a Balloon Note

1. Choose **Insert** > **Balloon**.
2. Choose the appropriate commands from the NOTE TYPES menu on the Menu Manager .
3. Choose **Done** and select the leader type, if necessary.
4. Select edges or the start point for the leaders; then choose **Done Sel**.
5. Pick the location of the balloon note and type the text.
   Note: You can use the pop-up menu to manipulate balloon notes the same way you would manipulate drawing notes.

Adding Balloon Notes

Balloon notes consist of text enclosed in a circle. You can add them freely anywhere on the drawing, or attach them to any number of edges. When you enter multiple lines of text for the balloon note, the system encloses each line within a balloon, stacking them horizontally or vertically next to each other, depending upon whether you choose **Horizontal** or **Vertical** from the NOTE TYPES menu. The size of these related balloons is equal to the largest one necessary to enclose the longest text.

To restrict the size of a balloon, use the drawing setup file options `max_balloon_radius` or `min_balloon_radius`.

To Wrap Note Text

When a note is created the text string is entered as one line. Use this procedure to wrap the text to a desired width.

1. Select the note. The text is highlighted and "handles" appear on either side of the string.
2. Use the cursor to drag the right side text area handle to the desired width. The text wraps according to the width of the text area.

If you add text to a note, the new text is not wrapped. To wrap it, you can either

- Re-drag the text box to the desired width, or

- Select the note and from the right mouse button pop up menu, click Wrap Text. The note is wrapped to the longest line.

To Create Superscripted and Subscripted Text

To create superscripted text, type `[@+text@#]` and to create subscripted text, type `[@-text@#]`, where `text` is the note text that is superscripted or subscripted.

You can superscript or subscript plain text and special symbols; however, you cannot super- or subscript dimensions, instance numbers, other parameter values, or geometric tolerances. You can create superscripted and subscripted text as a separate note or include it in a text line with regular text on either side of it, and then modify it independently of the rest of the text line in which it appears. The system positions the text in reference to the closest line of regular text, whether that text is part of another note, or part of the note currently being created.

To Place Draft and Reference Dimensions in Notes and Tables

You can place draft (add and dd) dimensions and reference (rd) dimensions parametrically in drawing notes and tables using &add or &dd. Draft (driven) dimensions and reference dimensions created in the drawing are updated when the model is regenerated.

To place draft dimensions and reference dimensions in drawing tables, choose **TABLE** > **Enter Text**. Select the table cell, and enter the symbol of the dimension, preceded by an ampersand (&).

To Enter Special Text Characters

When typing text strings into notes, you can use special text characters, special symbols, and drawing symbols.

You can add special symbols to text strings in notes by typing them from the keyboard, the palette window, or a text file. For a table listing all of the special symbols available with Pro/ENGINEER, as well as their definitions and ASCII character representations, see the *Pro/ENGINEER Installation and Administration Guide*.

Whenever you want to display the characters & or @ in the text of a note, you must type them twice. For example, type [`See Views 1 && 2 @@ Sheet 3`] for the note to read `See Views 1 & 2 @ Sheet 3`.

To Add Special Symbols to a Text Symbol Using the Keyboard

1. Press CTRL-a to turn on the special symbol font file.
2. Type the ASCII characters that represent the special symbols.
3. Press CTRL-b to turn off the special symbol font file.

The system displays the special Symbol Palette window whenever you begin to enter note text.

The Symbol Palette Input Configuration File Option

You can use operating system commands to resize the palette window, but the window reverts to its normal size each time that you press the ENTER key. If you set the configuration file option `symbol_palette_input` to `yes`, the palette appears whenever you begin to enter a note. If you set it to `no`, it does not appear at all.

As you move the pointer over each symbol or character on the palette, the system highlights that symbol; if you select it, it enters the symbol's ASCII equivalent (enclosed between ^A and ^B) in the text of the note you create. To define your own special fonts and symbols, see the *Pro/ENGINEER Installation and Administration Guide*.

To Add Drawing Symbols to Notes

You can include a drawing symbol in a note if its instance is present in the drawing. To include a symbol, use the following format:

```
&sym(<symbolname>)
```

Type the symbol filename without an extension.

> **Note:** You cannot call out drawing symbols in dimensions that are stored with the model (those of type `d` or `ad`).

When you include an instance of a generic symbol, after you type the symbol name, select groups that compose the instance (this is similar to creating instances of a generic symbol).

For example, if a drawing contains an instance of the symbol `bevel`, to include the symbol in a note, type [`&sym(bevel)`].

When you edit a note using the **Full Note** or **Text Line** commands, the system represents the symbol in the following format:

```
n:&sym(sym_path)
```

where `n` is the number of the text element, and `sym_path` is the name or pathname of the symbol.

Note: Pro/ENGINEER displays a drawing symbol in the text note to which it is added, but it does not display it in the dimension text line. Dimension text resides in Part mode; therefore, it may not acquire drawing symbols that reside in Drawing mode.

To Include the Symbol:
1. Choose **Insert** > **Note**.
2. Type the note in the format described earlier.

3. Specify the symbol height.
4. For an instance of a generic symbol, select groups to compose the instance. To complete the instance description, type variable text, if necessary.
5. To finish the note, press the ENTER key twice.

To Use the Geometric Tolerance Dialog Box

To access the geometric tolerance functionality in Drawing mode, click **Insert > Geometric Tolerance**.

Using the four tabbed pages of the **Geometric Tolerance** dialog box, you can perform the following tasks:

- Specify the model and the reference entity to which to add the geometric tolerance, as well as place the geometric tolerance on the drawing.
- Specify the datum references and material conditions for a geometric tolerance, as well as the value and datum reference of a composite tolerance.
- Specify the tolerance value and the material condition.
- Specify the geometric tolerance's symbols and modifiers, as well as the projected tolerance zone.

The left side of the dialog box displays the geometric tolerance symbols available for selection. The next table shows the geometric tolerance types and the appropriate entities that you can reference. When you place the cursor on one of the geometric tolerance symbols, the type appears in the dialog box as balloon help and in the message status line.

After you have created a geometric tolerance, click **New Gtol** to create additional geometric tolerances. After you have modified an existing geometric tolerance, click **New Gtol** to create a copy of the geometric tolerance that was just modified.

Notes:

Reference Entity Types

Not all reference entity types are available for all geometric tolerance types. You must select a new entity whenever you change the entity type. The system does not allow you to complete the creation of a geometric tolerance until you select a reference entity. The reference entity is the geometry or feature that the geometric tolerance controls; you should not use it in place of a set datum or as an attachment type for the gtol.

Removing GTOL Restrictions

Set the configuration option `restricted_gtol_dialog` to **no** to remove the standards restrictions mentioned above when picking certain GTOL types.

Example: Geometric Tolerance Classes and Types

Class	Type	Symbol	Reference Entity
Form	Straightness	—	Surface of revolution, axis, straight edge
	Flatness	▱	Plane surface (not datum plane)
	Circularity		Cylinder, cone, sphere
	Cylindricity		Cylindrical surface
Profile	Line	⌒	Edge

	Surface		Surface (not datum plane)
Orienta-tion	Angularity	∠	Plane, surface, axis
		∥	
	Parallelism	⊥	Cylindrical, surface, axis
	Perpendicularity		Planar surface
Location	Position	⊕	Any
		◘	
	Concentricity	≡	Axis, surface of revolution
	Symmetry		Any
Runout	Circular	↗	Cone, cylinder, sphere, plane
		↗↗	
	Total		Cone, cylinder, sphere, plane

Model Refs Options

- **Model**—Specifies the model in which to add the gtol. You must explicitly select the model first. You can select the top model from the **Model** list or click **Select Model** to select a submodel. By default, the current geometric tolerance model is the current drawing model.
- **Reference: To Be Selected**—Specifies the reference entity type (edge, axis, surface, feature, datum, entity, or none). Select an item from the **Type** list or click **Select Entity**.
- **Placement**—Places the geometric tolerance in the drawing. Click **Place geometric tolerance** and select an item from the **Type** list:
  - **Dimension**—Attaches the geometric tolerance to a dimension. The drawing setup file option `gtol_dim_placement` determines its location below the dimension. You can attach multiple gtols (stacked) to the same dimension, and they behave as a single tolerance when you manipulate the dimension symbol.
  - **Datum**—Attaches the geometric tolerance to a datum. If the geometric tolerance is already attached directly to a datum, the system selects this item for you.
  - **Leaders**—Attaches the geometric tolerance to multiple edges (including datum quilt edges) with leader lines or to dimension witness lines.
  - **Free Note**—Places the geometric tolerance anywhere on the drawing. In Part mode, you cannot display *free* gtols that were created in Drawing mode. You can use **Free Note** placement only when you have selected **The Drawing** from the **Model** list as the top model.
  - **Normal Leader**—Attaches the geometric tolerance to an edge along a leader line that is perpendicular to the selected edge.
  - **Tangent Leader**—Attaches the geometric tolerance to an edge along a leader line that is tangent to the selected edge, orienting the geometric tolerance text box at the same angle as the leader.
  - **Other Gtol**—Attaches the new geometric tolerance to an existing one (you cannot attach the existing geometric tolerance to a dimension).
  - **Dim Related**—Relates a geometric tolerance to specified dimension text. Select this item from the **Type** list; then select dimension text and a location relative to the text. To relate an existing free geometric tolerance to dimension text, choose TOOLS > **Relate Obj**.
  - **Make Dim**—Creates a driven dimension and attaches the dimension to it. This dimension belongs to the drawing. The geometric tolerance appears in standard dimension format, but with the geometric

tolerance instead of a dimension value.

Tip: Reference Entity Types

Not all reference entity types are available for all geometric tolerance types. You must select a new entity whenever you change the entity type. The system does not allow you to complete the creation of a geometric tolerance until you select a reference entity. The reference entity is the geometry or feature that the geometric tolerance controls; you should *not* use it in place of a set datum or as an attachment type for the gtol.

Datum Refs Options

- **Datum References**—Specifies the primary, secondary, and tertiary datum references for any geometric tolerance that permits datum references. Click **Primary**, **Secondary**, or **Tertiary** and select an item from the **Basic** and **Compound** lists. The lists contain the currently selected datum and all other datums in the current geometric tolerance model. If you want to select another set datum or axis, click **Select...**. For basic and compound datum references, you can also specify a material condition by selecting an item from the **Basic** and **Compound** lists.
- **Composite Tolerance**—Creates a composite gtol. Type a value in the **Value** box and select the datum references to show from the **Datum Reference** list.
- **Unordered**—Select this checkbox to allow two datum references to be listed in the same section of the control frame.

Example: Datum References for a Composite Tolerance

1   No datum reference.
2   Primary datum reference.
3   Secondary datum reference.
4   Tertiary datum reference.

To Specify the Tolerance Value and Material Condition

Using the buttons on the **Tol Value** page of the **Geometric Tolerances** dialog box, you can specify the tolerance value and material condition.

- **Tolerance Value**—Specifies the tolerance value. Type a value in the **Overall Tolerance** box, or select the **Per Unit Tolerance** check box and type a value in the **Value/Unit** and **Unit Length** boxes (or **Unit Area** for some geometric tolerance types).

- **Material Condition**—Specifies the material condition.

LMC	Ⓛ	Least material condition
MMC	Ⓜ	Maximum material condition
RFS	Ⓢ	Regardless of feature size
RFS/Default		RFS, but does not show a symbol in frame

To Specify Symbols and Modifiers

Using the buttons on the **Symbols** page of the **Geometric Tolerance** dialog box, you can specify the gtol's symbols, modifiers, and the projected tolerance zone.

- **Symbols and Modifiers**—Specifies the geometric tolerance symbols and modifiers. Select the **Statistical Tolerance**, **Diameter Symbol**, **Free State**, **All Around Symbol**, **Tangent Plane**, or **Set Boundary** check box.
- **Projected Tolerance Zone**—Specifies the location of the projected tolerance zone. Select **None**, **Below Gtol**, or **Inside Gtol**. You can also specify the height of the projected tolerance zone by typing a value in the **Zone Height** box.
- **Profile Boundary** (available for Profile gtols only)—Specifies unilateral direction, bilateral direction, or both. The Flip side allows you to modify the profile direction.

About Using Reference Datums

Before you can reference a datum plane or axis in a geometric tolerance, you must set it. When you set a datum, its name is enclosed in a rectangle. Once you have set a datum, you can still use it in the usual way to create features and assemble parts.

When displaying set datum planes, keep in mind the following:

- A set datum plane, attached to a dimension, appears in a drawing only when you *show* the dimension to which it was associated.
- The system does not show a reference datum plane in a drawing unless it is perpendicular to the screen.

To Set a Datum in Drawing

When you set a datum, you can also choose to attach it to a dimension.

1. Select the name of the datum you want to set.
2. Click **Edit** > **Properties**. The **Datum** dialog box opens.
3. In the Type field, click the rectangle-surrounded "-A-."
4. Use the **Placement** field to specify whether the datum is free or part of a selected dimension.
5. Click **OK**.

To Place a Set Datum in a Dimension

1. Select the name of the datum you want to place.
2. Click **Edit** > **Properties**. The **Datum** dialog box opens.
3. Use the **Placement** field to specify whether the datum is free or part of a selected dimension.
4. Click **OK**.

Note: If you set the drawing setup file option `gtol_datums` to `STD_ASME` or `STD_ISO`, you can also place a set datum on a geometric tolerance. Click **In geometric tolerance** and **Pick Gtol** in the **Datum** dialog box; then select a geometric tolerance in the model.

To Create a Reference Datum Attached to a Cylindrical Surface

1. Select the axis. **Note:** The axis must exist on the cylindrical view.
2. Click **Edit** > **Properties**. The **Axis** dialog box opens.
3. Click the rectangle-enclosed A in the **Axis** dialog box to change to ISO standard, and then select **On Geometry** in the **Placement** area.
4. Select the cylindrical surface to which you want to attach the reference datum. The reference datum snaps to the cylindrical surface.
   Note: The reference datum attaches at the point where you made the selection on the surface.
5. Click **OK**.

Example: Geometric Tolerance Symbol with a Compound Datum
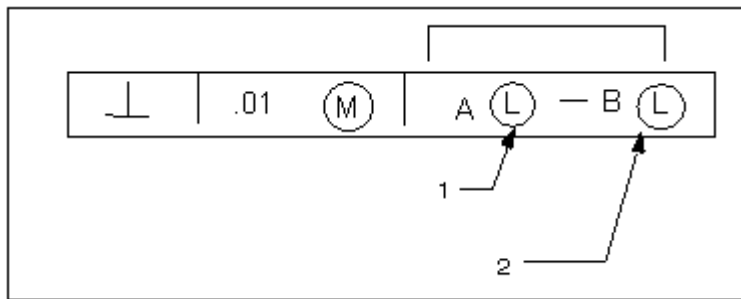
1   First datum feature.
2   Second datum feature.

About Basic Dimensions

Basic dimensions are theoretically exact dimensions that appear with the measurement value and any associated text enclosed in a feature control frame. If the original dimension has tolerances, the system automatically removes them; you cannot add tolerances to basic dimensions. You can transform existing dimensions into basic dimensions, and set inspection dimensions according to the DIN standard.

To Set A Dimension as a Basic Dimension

1. Select the dimension to change.
2. Click **Edit** > **Properties**..
3. In the **Display** field, click **Basic**.
4. Click OK.

To Set a Dimension as an Inspection Dimension

5. Select the dimension to change.
6. Click **Edit** > **Properties**..
7. In the **Display** field, click **Inspection**. The dimension is enclosed in an oval.
8. Click OK.

About Drawing Datum Targets

Use datum targets to indicate critical measurement points on the plot. The target is drawn inside a scalable circle with a leader. You can create a datum target specifying any set datum point, except one that has been created using **Offset Csys**. *Simple* datum targets reference a selected set datum point on a surface or edge.

Diameter datum targets contain a required diameter.

When a target is added, the lower part of the circle references the datum name, and adds an integer to note the instance of the target symbol; for example, DTM5 becomes DTM51, DTM52, etc.

Draft targets do not resequence themselves if you delete one of them. For example, if there are three datum targets (B1, B2, and B3), deleting B2 leaves B1 and B3, not B1 and B2.

Using the **Show/Erase** dialog box, you can show datum targets in multiple views.

To Create Datum Targets

1. Choose **Insert** > **Target**.
2. In the drawing, select a *set* reference datum. You can select draft datum planes and axes as well.
3. Using the TARGET menu in the Menu Manager, do one of the following:
   – Choose **Simple** to contain the name of the referenced datum in the lower half of the datum target symbol.
   – Choose **Diameter** to select a dimension point for each datum point (shown in the top half of the datum target symbol).
   – Choose **Line** to select a datum curve. The curve you select must be a line, and it must reside in the view where you selected the set datum plane or axis. You cannot select a datum curve that is not a line in this view.
4. Using the GET DTM PNT menu, do one of the following:
   – Choose **Select Point** to select a datum point in the drawing.
   – Choose **Create Point** to create a new datum point.
5. Click on the location point for the target.
The target and leader are added to the drawing. You can move or resize the target symbol.

To Modify the Material Condition

1. Select a geometric tolerance.
2. Click **Edit > Properties**. The **Geometric Tolerances** dialog box opens.
3. Click the **Tol Value** tab.
4. Select a material condition from the drop-down list. The system updates the geometric tolerance on the screen.
5. Click **OK**.

To Modify a Datum Reference to a Geometric Tolerance

1. Select a geometric tolerance.
2. Click **Edit > Properties**. The **Geometric Tolerances** dialog box opens.
3. Click the **Datum Refs** tab.
4. Do one of the following:
   – Add a new datum reference by converting an existing simple datum reference into a compound reference feature. Add a secondary or tertiary datum reference clicking either **Secondary** or **Tertiary**. Select a datum reference from the **Compound** list. Specify a material condition, if needed.
   – Replace a datum reference with another datum. Click **Primary**, **Secondary,** or **Tertiary** (whichever is appropriate) in the **Datum References** box; then select a different datum name from the **Basic** or **Compound** list. If the datum that you want to use is not on the list, click **Select...** to select it in the drawing.
   – Remove a datum reference. Click **Primary**, **Secondary,** or **Tertiary** in the **Datum References** box (whichever is appropriate); then select **None** from the **Basic** or **Compound** list.
5. Click OK.

Tip: Adding Geometric Tolerances to Notes

Every geometric tolerance symbol on a drawing has a symbolic representation. You can include a geometric tolerance in a note as a parameter by typing its *symbolic* value in the format [&g#], where "#" is the number indicating the order in which the geometric tolerance was created.

Example: Symbolic Representation

1   Geometric tolerance symbol.
2   Switched to symbolic format.

Example: Note Creation
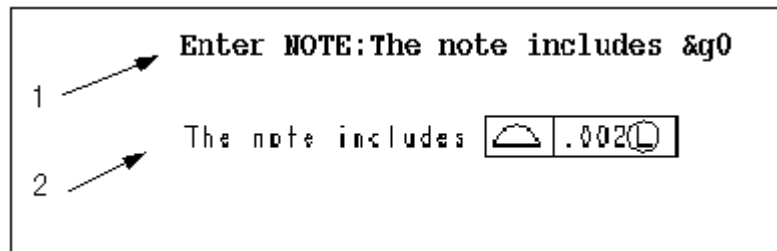
1   Type a note at the system prompt.
2   This note appears on the screen.

To Delete Geometric Tolerances

In Drawing mode, you can remove a geometric tolerance by selecting it and using **Delete** from the right mouse button pop up menu. However, a geometric tolerance remains intact in a note even if you delete it from the drawing. If you delete it from Part mode, the system replaces it in the note with "***".

About Showing Geometric Tolerances in Different Modes

You can create assembly, part, or draft (not associated with the model) geometric tolerances in Drawing mode. However, geometric tolerances created in Part and Assembly modes do not automatically appear in Drawing mode.

When you place a geometric tolerance on a dimension in Part mode, keep in mind the following rules:

- If you place a geometric tolerance on a dimension in a part, and then create a drawing using that part, you must show the dimension in the drawing first; otherwise, the geometric tolerance does not appear.
- If a geometric tolerance is attached to a part dimension that the system cannot display in an assembly drawing, it does not display the geometric tolerance either.

You can use the View > **Show and Erase** dialog box to display geometric tolerances that were created in Assembly and Part modes.

About Surface Finish Symbols

You can add surface finish symbols to a model using standard surface finish symbols available with Pro/DETAIL, or you can create your own surface finish symbols. However, you *cannot* attach surface finish symbols to parts intersected by an assembly feature.

If you have a license for Pro/DETAIL, you can access a set of standard surface finish symbols in the Pro/ENGINEER directory *loaddirectory/symbols/surffins*, as shown in the following table.

	Generic	Machined	Unmachined
no_value	√	∇	∜
standard	√ value	∇ value	∜ value

Surface Finishes for Drawings

Pro/ENGINEER reflects drawing surface finishes only in the drawing. It supplies a generic surface finish symbol that consists of building blocks, or groups. To create a desired instance, you must select any groups that you want to include in the symbol and specify required information. The generic surface finish symbol is located in the system symbols area.

Surface finishes are associated with surfaces in the part, not the entities or views in the drawing. Each surface symbol applies to the entire surface. When you specify a surface finish for a surface that already has one, Pro/ENGINEER redefines the surface finish information in the part and replaces the old symbol with the new one. Just as you cannot display the same dimension in two different views, in Pro/ENGINEER, you cannot display the same surface finish in two views.

If you create and add your own surface finish symbols, specify their location by setting the configuration file option `pro_surface_finish_dir`.

To Add Surface Finish Symbols

1. Choose **Insert > Surface Finish**.
2. Using the **GET SYMBOL** menu, do one of the following:
   – Use **Name** to select a symbol from the namelist menu. This menu lists all of the symbols that are currently in the drawing. When symbols appear to have the same names, but are, in fact, from different symbol directories, the system identifies them in the SYMBOL NAMES menu with a number in parentheses (#). The one-line help for each name provides the name and path for each symbol.
   – Use **Pick Inst** to select a symbol by selecting any instance of the symbol in the drawing.
   – Use **Retrieve** to select a symbol by choosing one from a list of symbols on disk. Selecting **UP** brings you up one directory. Selecting another directory name brings you down to that directory. The menu name is the name of the current directory. Unlike storing a symbol, you can retrieve one from anywhere you have read permission, both up and down the directory tree.
3. Choose one of the commands from the INST ATTACH menu.
4. Respond to the system prompts and menu commands for the commands you chose.
5. Type a value for the surface roughness. The system places the symbol on each surface you select on the drawing with the size relative to the default text height.

The INST ATTACH Menu

- **Leader**—Creates the symbol with a leader. Choose commands from the ATTACH TYPE menu .
- **Entity**—Attaches the symbol to an entity (model edge or draft geometry).
- **Normal**—Attaches the symbol to an edge, entity, or dimension. Positions it with the vertical reference pointing up. When attached to an edge at an angle, a yellow arrow indicates To position it normal to the
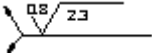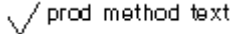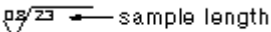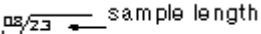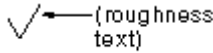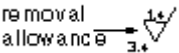
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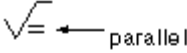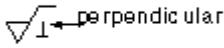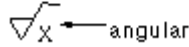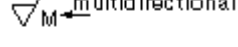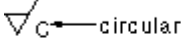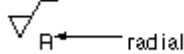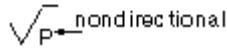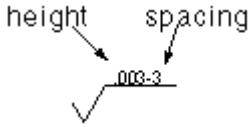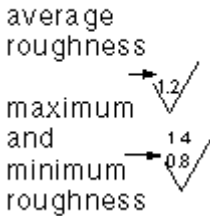
- **No Leader**—Creates a symbol that is unattached.
- **Offset**—Creates a symbol without leaders that is placed relative to a detail entity.

Valid Surface Finish Symbol Groups

The following table describes the groups that you can use to create a surface finish symbol. The first column indicates whether it is an ANSI or ISO symbol, or both.

Creating a Surface Finish Symbol

ISO/ ANSI	GROUPS	DESCRIPTION	ILLUSTRATION
ANSI ISO	LEADER	Used to create a symbol leader.	
ISO ANSI	MACHINED	Material removal by machining is required.	
ISO ANSI	NO_REMOVAL	Material removal prohibited.	
ISO only	PROD_METHOD	Text used for specifying production method.	
ANSI only	ROUGH_ SPACE	Required maximum roughness spacing (mm or inch).	
ISO ANSI	SAMPLE_LEG	Roughness sampling length or cutoff rating (mm or inch).	
ISO only	OTHER_ROUGH	Text used for specifying other roughness.	
ANSI only	DESIGNATION	Text used for special designations.	
ISO ANSI	REMOVAL ALLOW	Material removal by machining that is required to produce the surface (mm or inch).	

ISO ANSI	LAY	Designates direction of lay. The system supports the following types:	
		PARALLEL—Approximately parallel to the line representing the surface to which the symbol is applied.	
		PERP—Approximately perpendicular to the line representing the surface to which the symbol is applied.	
		ANGULAR—Angular in both directions to the line representing the surface to which the symbol is applied.	
		MULTI_DIR—Multidirectional.	
		CIRCULAR—Approximately circular relative to the center of the surface to which the symbol is applied.	
		RADIAL—Approximately radial to the center of the surface to which the symbol is applied.	
		NON_DIR—Nondirectional, or protuberant.	
ISO ANSI	UNSPECIFIED	Basic surface texture symbol. You can produce the surface by any method, except when the bar or circle is specified.	
ANSI only	WAVINESS	Specified waviness (mm or inch):	
		WAVE_HEIGHT— Maximum waviness height.	
		WAVE_SPACE— Maximum waviness spacing.	
ISO ANSI	ROUGHNESS	Roughness rating indicates permissible roughness range (mm or inch). The system supports these types:	
		AVERAGE— Roughness average rating.	
		MAX_MIN—Maximum and minimum roughness average values.	

About Drawing Tables

A drawing table is a grid of rows and columns in which you enter text. It has justified text, cells in which you type text, and a specified number of characters and lines per cell. The text in a drawing table has complete drawing text functionality; you can modify it using the MODIFY TEXT menu commands. You can enter dimension symbols and drawing labels as well, and the system updates them as you modify the model or drawing. You can include a drawing table in drawing formats, drawings, and layouts.

Using Pro/Report with Tables

The optional Pro/Report license lets you designate certain cells as "repeat regions" that automatically populate with design data that you specify through special report parameters. If you define these regions in a report file, (.rep) they are read-only. If you define them in a drawing however, you can edit values shown in the drawing table and pass the edits back to the associated parts.

Click *See Also* for more information on this subject.

Using the Pop-Up Menu

Using the Pro/ENGINEER pop-up menu, you can modify and manipulate drawing tables in the following ways:

- Move them to another location on the drawing sheet.
- Edit cell text and text style as you would a note, with one exception. When you choose a cell belonging to a repeat region, you can enter a Pro/REPORT symbol.
- Remove the contents of the cell or the entire table.
- Change the size of the row or column of the selected cell in either drawing units or text height/length units.
- Rotate the table 90 degrees about its origin.
- Specify a new origin for the table (you can only select a corner of the table segment of the active cell).

To Create a Drawing Table

1. Choose **DRAWING** > **Table** > **Create**.
2. From TABLE CREATE menu, choose a command.
3. Locate the table by clicking on the drawing. If you chose **Descending** and **Rightward**, the system uses the position to indicate the upper-left corner of the table.
4. A scale of numbers appears. The system displays these numbers according to the current text height in the drawing setup file. Mark off the width of each column (the number of characters that the row can contain at the current text height) by selecting a number. Choose **Done** when you have finished.
5. Mark off the height of each row (the number of lines of text that the row can contain at the current text height) by selecting a number. Choose **Done** when you have finished.

To Enter Text in a Table

To add text to or edit text in a drawing table,

1. Choose **Enter Text** from the **TABLE** menu.
2. Select a cell and type the text. You can type multiple lines.
   If you set the configuration file option `symbol_palette_input` to `yes`, a palette window appears listing special characters available with Pro/ENGINEER. To use the palette, first select a cell; then select characters from the palette.
3. To finish, press **Enter** twice.

When you add text to the table, the system sets its height according to the drawing setup file option `drawing_text_height`. To override the current setting, or to modify the parameters of the selected table text, use the **Mod Text Style** command on the right mouse button pop up menu.

To Word Wrap Drawing Table Text

1. Select the table cell containing the text

2. From the right mouse button pop up menu, click **Wrap Text**. The text wraps to the column width.
Note: If you need to extend the row height, use the Mod Row Height command on the right mouse button pop up menu.

Tip: Tables on Layers

You can place drawing tables on a layer by choosing **Dwg Table** from the LAYER OBJ menu. Text in the table has the same display status as the table.

To Copy Cell Contents

You can copy the contents of a cell into another (empty) cell or create a copy of the entire table in the drawing.

1. Choose **Drawing > TABLE > Copy > Copy Cell**.
2. Select a cell to copy.
3. Select destination cells. The system copies the contents of the highlighted cell. Click the middle button to exit.

To Copy a Table

1. Choose **Drawing > TABLE** > **Copy** > **Copy Table**.
2. Click inside the table to highlight the original.
3. Using a command in the GET POINT menu, select a point on the screen to locate the origin of the new table.

To Merge the Cells of a Table

1. Choose **Drawing > TABLE > Modify Table > Merge**.
2. From the TABLE MERGE menu, choose **Rows** and **Columns** or **Rows & Cols**.
3. Select the opposite corners of the range of cells to convert by selecting within the cells.

Restrictions When Merging Cells

When merging cells, keep in mind the following restrictions:

- The columns or rows you are merging must be similar.
- The cell containing text must be positioned properly with respect to the origin of the table. For example, in a rightward, descending table, a cell with text must be at the left (upper) end of the range of cells you are going to merge, with the empty cell at the right (lower) end of the range.
- Only one cell in the range can contain text.

To Remesh the Rows and Columns of a Table

1. Choose **TABLE > Modify Table > Remesh**.
2. Select the range of cells to remesh by selecting the opposite corners of the range of cells. Only the first cell in the range can contain any text.
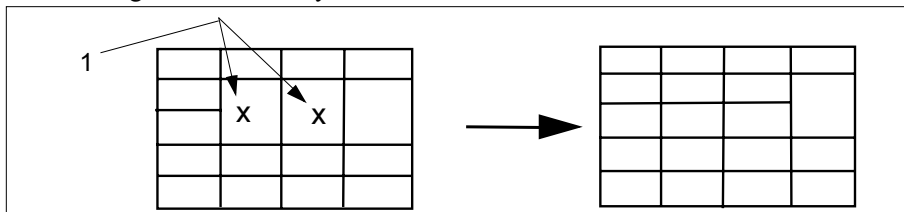
1   Pick here to get the resulting table.

To Change the Origin of a Table

1. Choose **TABLE > Modify Table > Origin**.
2. Locate the fixed corner of the table by selecting within the corner cell. The system changes the table origin. The default table origin is the upper-left corner. The specified corner now remains stationary when you add rows or columns to the table.

To Rotate a Table 90 Degrees

1. Choose **TABLE > Modify Table > Rotate**.
2. Select inside the table. It rotates by 90 degrees in the counterclockwise direction about its origin.
3. Continue selecting inside the table; each time that you select, the table rotates 90 degrees.

To Blank or Display Lines in a Table

1. Choose **TABLE > Modify Table > Line Display**.
2. From the TABLE LINE menu, choose one of the following commands:
   - **Blank**—Blanks the selected line segment.
   - **Unblank**—Redisplays the selected line segment in the table.
   - **Unblank All**—Redisplays all blanked lines in the table.

Tip: Blanking a Line at the End of an Element

In a region, when you attempt to blank a line at the end of an element that falls between elements of the same range, you must confirm that you want to blank the selected line even at the end of the range.

Modifying the Line Font, Color and Width

You can modify the line font, color, and width of a table grid the same way you would modify geometry. The changes apply to all grid lines in the table.

To Insert a Row or a Column

1. Choose **TABLE > Mod Rows/Cols > Insert**.
2. Do one of the following:
   - Choose **Row** and select a horizontal line in the table. The system displays a new row between the two rows that border the selected line.
   - Choose **Column**; then select a vertical line in the table. The system inserts a new column between the two columns that border the selected line.

To Remove a Row or a Column

1. Choose **TABLE > Mod Rows/Cols > Remove**.
2. From the ROW/COL OPTS menu, choose **Row** or **Column**.
3. Select inside the row or column to remove.

To Resize Rows and Columns

1. Choose **TABLE > Mod Rows/Cols > Change Size**.
2. From the ROW/COL OPTS menu, choose **Row** or **Column**.
3. Select inside the row or a column.
4. Type the new height for the row or the width for the column.

Working With Rows and Columns

Using the **ROW**/**COLS OPTS** menu, you can manipulate rows and columns in a table in the following ways:

- Insert a row or a column.
- Remove a row or a column.
- Change the height of a row or a column.
- Modify the justification of cell text.

You can also manipulate multiple rows or columns at one time by continuing to select cell borders after the first selection is made.

To Justify Text in a Column

1. Choose **TABLE** > **Mod Rows/Cols** > **Justify**.
2. From the **HOR JUST TXT** menu, set the horizontal justification.
3. Set the vertical justification.
4. Select columns to justify. The system justifies any text you add subsequently, as you specified.

Note: To clear a table cell justification after typing text, click **File** > **Delete** and select the text.

Horizontal and Vertical Justification Options

- **Left**—Left-justifies the text.
- **Center**—Centers the text.
- **Right**—Right-justifies the text.
- **Hor Default**—Horizontally justifies the text using the default setting (left).
- **Top**—Justifies the text to the top of the cell.
- **Middle**—Justifies the text to the center of the cell.
- **Bottom**—Justifies the text to the bottom of the cell.
- **Ver Default**—Vertically justifies the text using the default setting (top).

To Modify the Justification of Individual Table Text

1. Select the cell to modify.
2. From the right mouse button pop up menu, click **Mod Text Style**.
3. In the dialog box, do one of the following:
   - Specify the horizontal justification by selecting a value from the **Justify Horiz** list.
   - Specify the vertical justification by selecting a value from the **Justify Vert** list.
4. Click **Apply**. The system changes the justification of the text as you specified.

To Move a Table Within a Single Sheet of a Drawing

1. Choose TABLE > **Move**.
2. Select one of the corners of the table to move.
3. Select a new position for that corner.
4. To move a table to another sheet, choose **Switch Sheet** from the SHEETS menu.

To Delete a Table

1. Choose TABLE **> Delete**.
   Note: This command deletes the entire table, *not* just a row or column.
2. Select the table to delete.
3. Type [Y] to confirm.

Storing and Retrieving Tables

Using the **Save/Retrieve** command in the TABLE menu, you can store a table and then retrieve it into any drawing. You can store or retrieve a table in Drawing, Layout, Format, Report, and Diagram modes. If you

retrieve a table into a drawing that is not the same size as the drawing in which it was stored, it retains its absolute size, that is, the size that it was when it was created.

To Store a Drawing Table

1. Choose TABLESTORE > **Store**.
2. Select anywhere within a drawing table.
3. Type a name for the table. The system saves it to a file in your current directory and informs you that it has saved `tablename.tbl`. It gives the filename a unique version number to prevent overwriting tables with the same name.

To Store the Table Text

1. Choose TABLE > Save/Retrieve > Store Text.
2. Select the table.
3. Type the name of the file (without an extension) where you want to store the table text. The system creates a file with the name `filename.1`. It increments the file extension by 1 whenever you specify a name that another text file in your directory already uses.

Tip: Column Width

The location of text entries in the text files corresponds to that in the drawing table except when column strings exceed column width. When column entities have more characters than the column width allows, the file output expands the column width to display all characters in the string.

To Retrieve a Saved Drawing Table into the Current Drawing

1. Choose TABLE STORE > **Retrieve**.
2. Type the name of the table or type [?] to select the table from a namelist menu of stored tables.
3. The table's outline appears with its upper-left corner attached to the pointer. Place the table by selecting the location for the upper-left corner.

Tip: Listing Tables

Using the **List** command in the TABLE STORE menu, you can display an information window listing all tables stored in the current working directory, `pro_format_dir`, and `sys_format_dir`.

About Markups

A markup is an object that, like a set of transparent sheets on top of a drawing sheet, enables you to superimpose text and sketched entities in a variety of colors to indicate where changes may be required.

A markup is an informal sketch that you can create within Pro/ENGINEER with text superimposed over any object.

The basis for the markup is the object that you select to mark up. To create a markup, you use an object in the mode in which it was created:

- Part (includes all objects with a .prt extension).
- Assembly (includes all objects with an .asm extension).
- Drawing
- Manufacturing
- Layout
- Report

- Diagram

The object accompanies the markup file, which uses the file extension .mrk. **Note**: The **Orientation**, **Model Display**, and **Advanced** commands are not available in the Pro/ENGINEER **View** menu after you create a new markup object. You must orient your three-dimensional object before you create it. The system saves the orientation with the markup.

Each markup is like a separate transparent sheet laid over the object and contains sketched entities and/or notes in a single color.

During the markup process, the system does not change the object in any way, and saves all sketched entities and notes created in Markup mode with the markup file, not with the object. Since the driving object does not change, the system does not store it when it stores the markup.

You can also create a markup for a multisheet drawing, continuing over all of the sheets.

Note the following rules of operation for Markup mode:

- With assemblies, the following applies:
  - When you create an exploded view for a markup item, it does not affect the exploded status of the views in other items.
  - You can simultaneously see only those markup items that have the same explode status, explode dimensions, and view orientation.
  - You can modify explode dimensions by using **Mod Explode** in the ENTER MARKUP menu.
- The system displays three-dimensional objects with their colors as well as drawing colors.
- A markup file must reside in the directory of the object that the markup references.
- Layout dimensions appear as symbolic.

When doing a comparison with three-dimensional markup items using the **Markup**, **Setup**, and **Show** commands, you can toggle between **Seen** and **Unseen** only with those items that have exactly the same view as the current markup item.

To Create a Markup

1. Choose **File** > **New**.
2. In the **New** dialog box, click **Markup** and **OK**.
3. From the **Markup** dialog box, select or retrieve an object.
4. Type a name for the object you are creating. The default name is your user ID.
5. Choose any of the following commands from the MARKUP menu to mark and annotate the object:
   - **Setup**—Changes the color of the markup, creates a plot of it, toggles its display, or changes to another sheet of the markup and drawing. The MRKP SETUP menu displays the following commands:

     Color—Changes the color of the markup. The default markup color is red.
     Show—Lists the markups in the current working area along with their color. A check mark appears next to the markup's name and color if it is currently displayed. Choosing the markup name again removes the check mark and turns off its display.
     Switch Sht—Displays an abbreviated SHEETS menu, enabling you to view and mark other sheets of the drawing.
     Text Height—Sets the text height before creating notes.
     Line Width—Sets the width of sketched entities before creating them.
   - **Note**—Places a note without a leader.
   - **Arrow**—Sketches an arrow. Use the left mouse button to select the point to which the arrow points; then select the point for the other end of the arrow. Press the middle button to abort arrow creation.
   - **Curve**—Selects the points of a curve to which a spline is to be fitted. Use the left mouse button to select the point, the middle button to stop the curve, and the right button to abort curve creation.
   - **Sketch**—Sketches a curve without selecting points for it. Hold the left mouse button down to create a free curve. Release the left button to end it.
   - **Line**—Creates a line using the left mouse button.

- **Move**—Moves a note, curve, or line.
- **Modify**—Changes the note text or line width of a markup item.
- **Delete**—Deletes a markup item (note, arrow, curve, sketch, or line).

Note: The **Note**, **Arrow**, **Curve**, **Sketch**, and **Line** commands are "sticky": once you have chosen one of these commands, you can continue creating that type of entity until you choose another command.

To Modify the Markup

To modify the markup, choose **Modify** from the MARKUP menu, followed by **Note** or **Line Width**.

About Working with Multimodel Drawings

You can manipulate views of different models that appear in the same drawing in several ways:

- Display an assembly and all of its component parts.
- Clearly show all part and assembly dimensions in the same drawing.
- Display several members of the same family with different sets of features.
- Work with detail items, tables, repeat regions, and so on, belonging to any one of the models.

When working with multimodel drawings, keep in mind the following:

- You must add all models that are shown in the drawing to the drawing before you can display them as views.
- The system sets the drawing scale of each model independently (you may notice the scale value at the bottom-left corner changing when you set different models). Modify this scale for each model when the model is active.
- To add views of a particular model, you must set that model as active.
- When you add relations, the system adds them to the current model. Therefore, select a model before adding a relation.
- When you replace a generic assembly with an instance assembly that uses PART_B instead of PART_A, you must choose View > **Show and Erase,** and click **Dimension** in the Show/Erase dialog box to show dimensions of PART_B.

To Add Models to the Drawing

Before you can use a model in a view, you must add it to the drawing. Adding a model to the drawing does not place a view of the model on the sheet, but lets the drawing reference the model so that you can place a view. To add the first model click DRAWING > **Views** from the menu and enter the model name. For all subsequent models, click DRAWING > **Views** > DWG MODELS > **Add Model**.

- You can remove a model by choosing **Del Model** from the DWG MODELS menu; however, you must delete *all* views of the model first.
- You can replace one part with another by choosing **Replace** from the DWG MODELS menu *if* the parts are members of the same family table.

Setting a Default Model

Pro/ENGINEER considers only one model in the drawing to be the default, or current, model. The last model added to the drawing becomes the current model, but you can set any model as current by choosing **Set Model** from the DWG MODELS menu.

The system references an active model when you are adding views, or for any function that requires a default model. For example, when you add a repeat region to a drawing table, type report parameters in the repeat region, and update the table, it lists information from the model that was the current drawing model at the time that you added the repeat region to the table. In addition, the Pro/ENGINEER **Info** menu and the **Regenerate** command in the DRAWING menu apply to the active model.

To Set a Model as Current

1. Choose **VIEWS** > **Dwg Models** > **Set Model**.
2. Do one of the following:
3. Select the model with which you want to work from the namelist menu. It contains only model names that you have added to the drawing.
4. Select a view of the model that you want to make active.
5. To highlight the active model, choose **DWG MODELS** > **Model Disp** > **Hilite Cur**.
6. To return the model display back to normal, choose **MODEL DISP** > **Normal**.

Note: The Set Model toolbar button is available as a shortcut to set the current drawing model and is included in the Toolbar Customization dialog box, accessed from the Pro/ENGINEER **Utilities** menu.

To Merge Drawings

Using the **Merge** command in the ADV DWG OPTS menu, you can merge two drawings. Merging allows increased performance and management of large drawings. Individual Pro/ENGINEER users can work in parallel and then merge their separate drawings into a single drawing file. A source file is appended to the target file as additional sheets; for example, when you merge a two-sheet source drawing into a four-sheet drawing, the target drawing then contains six sheets. You can later use SHEETS > **Reorder** to rearrange sheet order.

Note: Merging removes the source file from memory. If you previously saved the source file to disk, it is removed from session but still exists on disk.

1. Click **File** > **Open** and select and retrieve the target drawing or report file. The target drawing is the drawing into which you will merge another drawing.
2. Choose DRAWING > **Advanced** > **Merge**.
3. Use the Open dialog box to select the source file to merge into the current drawing. When you select a file, the source file is merged into the current (target) file.
4. Select additional source files to be merged, one file at a time, to add additional sheets to the current drawing.

Rules for Merging Drawings

The following rules and restrictions apply to merging drawings or reports:

- The target drawing file must be in session.
- A drawing or report cannot be merged into itself.
- The source drawing is removed from memory after it is merged.
- The main drawing file can contain one or more formats. Each sheet can use a different format. When a source drawing is merged into a target drawing that has a different format, the source format is not automatically replaced. You can replace the source format after the merge.
- When the source drawing to be merged contains dimensions that are the same as those of the target drawing, Pro/ENGINEER automatically removes these duplicate dimensions from the source drawing.
- When files to be merged reference a different version of the same model, the latest version in memory is used for the drawing.
- The source drawing and the target drawing must use the same drawing units.
- When the source drawing and the target drawing contain a model with the same name, even if these are different models, the model currently in session is used in the final drawing.
- After a file is merged, the drawing setup file options from the target drawing override the setup options used by the source drawing.
- If the source drawing contains a drawing program, it is deleted during the merge. To preserve a drawing program, use the drawing that contains the drawing program as the target drawing.
- Drawing parameters in the source drawing are deleted during the merge.

About Drawing Representations

Using drawing representations, you can improve performance, particularly when working with multimodel large assembly drawings and drawings of complex models. Drawing representation functionality is available only with an Advanced Assembly license.

A drawing representation is a series of commands that specifies a display configuration for the current drawing. You can use drawing representations to control which models and which views of a drawing the system retrieves and displays. For example, you can temporarily remove all models and views that are not necessary for current work.

With drawing representation functionality you can perform the following operations:

- Improve drawing retrieval time and improve drawing performance by minimizing display calculations such as view regenerations and repaints
- Configure the display of a drawing so that you can work with just the drawing information that you need for current work and keep unnecessary data out of memory
- Turn off the display of views that you are not working on so that the system does not calculate unnecessary display information and, in particular, does not retrieve models into session when you retrieve a drawing
- Load a drawing to a particular drawing sheet, zoom location, or view center—for example, retrieve only specified sheets of a drawing without loading all the sheets and displaying every view on each sheet

To Create a Drawing Representation

You can create a drawing representation using one of two methods:

- Create a drawing representation while you retrieve a drawing

  When you select a drawing to be retrieved, you can proceed as follows:
  - Access an existing drawing representation.

    After you retrieve the drawing, you can execute a drawing representation. The drawing and the drawing representation are displayed, and the drawing is configured according to the drawing representation.
  - Access the Drawing Rep Tool dialog box.

    After you specify the drawing to be retrieved, click **Open Rep** and then click **Create New Drawing Rep**. The Drawing Rep Tool dialog box opens. You can now create a new drawing representation. This method allows you to retrieve a drawing without bringing unnecessary views and models into session.
- Create a drawing representation after you retrieve a drawing

  When you already have a drawing in session, you can use the **DRAWING > Advanced > ADV DWG OPTS > Drawing Rep > Create** command to access the Drawing Rep Tool dialog box and create a drawing representation for the current drawing.

You give each drawing representation a name and execute it. When you save the drawing, the system also automatically saves the drawing representation for that drawing.

You can also use the DRAWING REP menu to copy or redefine a drawing representation that you previously created.

To Configure a Drawing Representation

You can configure a drawing representation to control view display and drawing display in the following ways:

- Display or erase all views
- Display or erase a particular view
- Display or erase views of a particular type, for example, all cross-sectional views
- Display or erase all views associated with a particular model
- Display or erase views on a specified sheet

- Load the drawing to a specified sheet
- Load the drawing to a pan/zoom state
- Load the drawing to the center of a specified view
- Set tables to frozen or updating

A drawing representation is never active—it is always executed. The drawing is always in a stage of configuration and can be changed at any time. Creating a drawing representation means that you are specifying one or more commands to configure the current drawing. As you specify commands, you can apply each command to the drawing immediately, or you can define several commands and then run them at once. A delay option is provided to allow you to configure the drawing without retrieving data until the configuration is complete.

The order in which you define commands is important. The system compiles a list of commands and runs them in the order in which you define them. The commands interact with each other cumulatively. For example, you can turn all views off (command #1) and then turn on all general views (command #2).

Drawing representations behave differently than simplified representations in Part and Assembly modes in that if one representation is executed after another representation, the drawing will display the results of both representations. For example, you could erase all views on sheet #1; then erase views on sheet #2; and then display all views. Each command is applied and the last command, when implemented, counteracts the earlier commands.

The DRAWING REP Menu

Use the DRAWING REP menu commands together with the Drawing Rep Tool dialog box to create and work with a drawing representation.

When you first access the menu, only the **Create**, **Execute**, and **Done-Return** commands are available.

After you have created a drawing representation, the other commands are also available.

The DRAWING REP menu contains the following commands:

Create—Opens the Drawing Rep Tool dialog box and allows you to configure a drawing representation for the current drawing. The dialog box automatically fills in the appropriate commands and updates the selections to the current state of the drawing.

Execute—Opens the Select Rep dialog box and allows you to select a rep to execute from the list of existing representations. Select a rep to execute, and click **OK**. The system executes the selected rep for the current drawing. That is, it executes the commands required to configure a drawing as defined by the drawing representation. To save the drawing representation, you must save the drawing.

Copy—Copies the selected user-defined drawing representation to a new drawing representation with an updated name.

Redefine—Displays the selected user-defined drawing representation in the Drawing Rep Tool dialog box so that you can modify it.

Delete—Deletes the selected user-defined drawing representation.

Info—Displays a printable window containing information about the selected rep.

Done-Return—Exits the DRAWING REP menu and returns to the ADV DWG OPTS menu.

The Drawing Rep Tool Dialog Box

The Drawing Rep Tool dialog box is divided into two pages and opens with the View Display page visible. The **View Display** page allows you to control view display configurations, and the **Drawing Display** page allows you to control sheet, pan/zoom, and table configurations. The following options are available for both pages:

Name—Displays a default name for the drawing representation that you can change

Navigate Sheets—Allows you to navigate through the sheets of a drawing to display the current sheet

OK—Accepts all the drawing configuration settings, closes the Drawing Rep Tool dialog box, and returns to the DRAWING REP menu

Execute—Executes the active command or applies all the listed commands to the drawing. When there is no active command, **Execute** is not available as an indication that all listed commands have been applied.

Default Drawing Representations

The system automatically creates two default drawing representations named All Views and No Views for all newly created drawings.

To create the default drawing representations for a previously saved drawing, retrieve the drawing and save it. The All Views and No Views default drawing representations are created.

* **All Views**—Turns on all views to display the full drawing, keeps tables updating, and resets the pan/zoom setting.

   All the models are retrieved.
* **No Views**—Turns off all views on all sheets of the drawing, keeps tables frozen, and resets the pan/zoom setting.

   No model is in session.
   You can use a No Views rep to create detail items such as notes and symbols that have no attachment to the model. You can retrieve the drawing showing only the notes and work on them without loading any models.

To Copy a Drawing Representation

After you have saved a drawing representation, you can copy the rep, using the **Copy** command on the DRAWING REP menu.

1. Choose DRAWING > **Advanced** > **Drawing Rep** > **Copy**. The Select Rep dialog box opens, listing the saved representations.
2. In the **Select Rep** dialog box, select a rep to copy from the list of existing representations, and click **OK**. You can copy only user-defined representations.
3. The system copies the selected drawing representation and opens the **Drawing Rep Tool** dialog box. The drawing representation name is automatically updated, the dialog box is populated with the same commands as those of the source rep, and the current commands are displayed in the command list area.
4. Rename and modify the new drawing representation.

To Redefine a Drawing Representation

After you have saved a drawing representation, you can redefine the rep, using the **Redefine** command on the DRAWING REP menu.

1. Choose DRAWING > **Advanced** > **Drawing Rep** > **Redefine**. The **Select Rep** dialog box opens, listing the saved representations.
2. In the **Select Rep** dialog box, select a rep to redefine from the list of existing representations, and click **OK**. You can redefine only user-defined representations.
3. The system opens the selected rep in the **Drawing Rep Tool** dialog box.
4. Redefine the selected rep.

To Delete a Drawing Representation

After you have saved a drawing representation, you can delete the rep, using the **Delete** command on the DRAWING REP menu.

1. Choose DRAWING > **Advanced** > **Drawing Rep** > **Delete**. The **Select Rep** dialog box opens, listing the saved representations.
2. Select a rep to delete, and click **OK**. You can delete only user-defined representations.

3. The system deletes the selected rep.

To Obtain Information About a Drawing Representation

After you have saved a drawing representation, you can obtain information on screen about the rep, using the **Info** command on the DRAWING REP menu

1. Choose DRAWING > **Advanced** > **Drawing Rep** > **Info**. The **Select Rep** dialog box opens, listing the saved representations.
2. In the **Select Rep** dialog box, select a rep from the list of existing representations, and click **OK**.
3. An information window displays such information as the name of the drawing representation, the commands that control view display, the commands that control the loading of the drawing. Choose **File** > **Save** to write the information to a file named drawingrep.inf, or choose **File** > **Save As** to write the information to a designated file.

To Create a New Drawing Representation While Retrieving a Drawing

You can create a new drawing representation upon retrieving a drawing. This procedure allows you to go straight to the Drawing Rep Tool dialog box in the No Views drawing representation state. No drawing models are retrieved, and **Delay View Command(s) Until Execute** is selected by default.

> Note: **Delay View Command(s) Until Execute** is selected by default for creating a drawing representation upon retrieval. This is a safeguard against accidentally retrieving the models into session while defining the drawing representation.
> With **Delay View Command(s) Until Execute** selected, you can specify several commands to configure the rep before executing any of them. Each command is added to the command list. The commands are not executed, and the system does not retrieve data, until you click **Execute**.
> When the **Delay View Command(s) Until Execute** check box is cleared, each command is executed immediately.

1. Choose **File** > **Open**.
2. Select a drawing from the File Open dialog box (click just once to select the drawing, not to open it) and click **Open Rep**. The Open Rep dialog box opens, listing the two default drawing representations, All Views and No Views.
3. Do not select a drawing representation, but select the **Create New Drawing Rep** check box, and click **OK**. The Drawing Rep Tool dialog box opens.
4. **Note:** Selecting **Create New Drawing Rep** is a convenient way to access the Drawing Rep Tool dialog box without retrieving any models. The system retrieves the selected drawing, with none of its views displayed and its tables set to frozen. The models are not retrieved into session.

To Execute a Drawing Representation While Retrieving a Drawing

You can execute a drawing representation directly into session while retrieving a drawing, using **Open Rep** in the File Open dialog box.

1. Choose **File** > **Open**.
2. Select a drawing from the File Open dialog box (click just once to select the drawing, not to open it) and click **Open Rep**. The Open Rep dialog box opens, listing previously saved drawing representations and the two default drawing representations, All Views and No Views.
3. Select the name of the drawing representation you want to execute, and click **OK**.
4. The system retrieves the drawing and configures the drawing to the selected drawing representation state.
5. **Note:** If the drawing is already in session and you close the drawing and then access the Open Rep dialog box, clicking **Cancel** brings up the drawing, which defaults to its previous state (either retrieves or does not retrieve models).

To Create a Drawing Representation for the Current Drawing

After you retrieve a drawing, you can create a new drawing representation for the drawing, using ADV DWG OPTS > **Drawing Rep** > **Create**.

1. Retrieve a drawing.
2. Choose DRAWING > **Advanced** > **Drawing Rep** > **Create**. The Drawing Rep Tool dialog box opens with the **View Display** page visible. The dialog box automatically fills in the appropriate commands and updates the selections to the current state of the drawing. The current commands are displayed in the command list area.
3. The **View Display** page can be used to configure view display.
4. In the **Drawing Rep Tool** dialog box, type a drawing representation name in the **Name** box or use the default.
5. In the Command Definition area of the dialog box, select **Display** or **Erase** to change the view display:
   - **Display**—Turns on the display of specified drawing views
   - **Erase**— Turns off the display of drawing views

     An erased view is replaced by a green outline containing the name of the view. To prevent erased views from being highlighted, clear **Highlight Erased Views** in the Environment dialog box.
     You can switch between **Display** and **Erase** as you specify cumulative commands.
6. Change the selection of views:
   - Select **All** to select all the views in the current drawing.
   - Select **Individual** to specify a particular view in the current drawing. The selection tool is automatically activated, and the GET SELECT menu appears, allowing you to select an on screen view. You can use **Navigate Sheets** to navigate to select a view on another sheet. When you select a view, its name appears in the box.
   - Select **Sheet** to specify the sheet(s) on which to apply the commands. Type the sheet numbers in the sheet box. Enter a sheet number or multiple numbers separated by a comma (1,3), or specify a range (5-12).
   - Specify views of a particular type by selecting **Type**, and then select a view name from the **Type** list.

   The **Type** list includes the most frequently used view types:
   - General
   - Projection
   - Detailed
   - Auxiliary
   - X-section
   - Broken
   - Partial
   - Exploded

   You can select more than one type in any applicable combination to refine the selection. For example, select Projection to select all projection views; then select X-section to specify only cross-sectional projection views.

   Select **Individual** to select an unlisted view type such as Graph, Revolved, or Of Flat Ply.
   - Select **Model** to select all views associated with the specified drawing model. Click the selection tool and select a drawing model from the list of available models (models belonging to the current drawing).
7. Each command that you define is listed in the command list area. When a rep is executed, all the commands are performed in sequential order, as they were entered. Control the command list as follows:
   - After you select an option to configure the drawing, click **Add** to add the command to the command list. **Add** has the same effect as **Execute** unless the **Delay View Command(s) Until Execute** check box is selected.
   - To insert a command into the command list, select a command in the list, then select an option, then

click **Insert**. The new command is inserted into the command list above the selected command. When the **Delay View Command(s) Until Execute** check box is selected, **Insert** does not execute until you click **Execute**.

- To delete a command from the command list, select a command and click **Delete**. The command is removed from the command list. When the **Delay View Command(s) Until Execute** check box is selected, **Delete** does not execute until you click **Execute**.

8. At any time, or when all the commands are listed to configure the drawing representation, click **Execute**. The system applies all the commands listed, in the order in which they are listed, to the current drawing. When all listed commands have been applied, **Execute** is not available unless **Delay View Command(s) Until Execute** is selected.

9. Click the **Drawing Display** tab.
The **Drawing Display** page can be used to automatically retrieve a specific sheet, go to a previously named view, or center a specific view. It also controls the behavior of tables with repeat regions.
Use the **Drawing Display** options to control the cosmetic components of a drawing. In the Drawing Location area of the dialog box, do one of the following:

- Select from the **Go to Sheet** list of available sheets in the current drawing to specify the sheet where to load the drawing representation (for example, load to sheet 6 of a 10-sheet drawing). When you retrieve a drawing representation, the specified sheet of the drawing is displayed. Only specified sheets are loaded when you retrieve the drawing representation.

- To specify a named pan/zoom location for the drawing, select the name of a view from the **Go to Named Pan/Zoom State** list. This list contains the same view names that are on the **View** > **Saved Views** list (the views that you previously defined).

When you next retrieve the drawing representation, the drawing zooms to the selected view. This allows you to make a minor change in a particular small part of the drawing without having to see the entire drawing.

- Select **Go to Center of View** to specify the location for the drawing representation. Centering a view keeps the drawing from repainting other views. Selecting **Go to Center of View** automatically clears **Go to Sheet** and **Go to Named Pan/Zoom State**.

    Select the view to go to from the list of views and models, or use the GET SELECT menu to select a view, and select the middle of the view. When you next retrieve the drawing representation, the drawing centers the specified view.

10. In the Table Preferences area, specify table control:
- **As Is**—Leaves BOM tables in their current state

- **Frozen**—Freezes BOM tables—repeat region calculations are frozen so any changes to the BOM assembly will not be reflected in the table

- **Updating**—Updates BOM tables—repeat regions automatically calculate changes to BOM assembly information

11. When you have finished configuring the drawing representation, click **OK** to accept the commands. The DRAWING REP menu appears.
After you have created a user-defined drawing representation, the entire DRAWING REP menu appears, and you can execute, copy, redefine, or delete a drawing representation, or display information about it in a printable window.

12. Save the drawing.
Note: Saving the drawing also saves the drawing representation.

Tip: Erasing Views by Menu or by Drawing Rep Tool Dialog Box

The Drawing Rep Tool Erase and Display options are functionally distinct from the Erase View and Resume View menu commands so that you can prevent previously erased views from reappearing when a drawing is retrieved.

When you erase a view using the Erase option in the Drawing Rep Tool, you must use the Display option in the Drawing Rep Tool to display the view again; any views that you erased using the menu command Erase View remain erased.

When you erase a view using the Erase View menu command, you must use the Resume View menu command to display the view again.

To Execute a Drawing Representation

You can execute a new, All Views, or No Views drawing representation. An All Views rep configures the drawing to turn on all views to display the full drawing, keeps tables updating, and resets the pan/zoom setting. A No Views drawing representation configures the drawing to turn off all views on all sheets of the drawing, keeps tables frozen, and resets the pan/zoom setting. When you load the drawing, no model is loaded.

1. Choose DRAWING > **Advanced** > **Drawing Rep** > **Execute**. The **Select Rep** dialog box opens.
2. Select the name of the drawing representation, and click **OK**. The system executes the selected drawing representation.

About Verifying Differences Between Drawings

Using the **Picture** command in the Pro/ENGINEER **Utilities** menu, you can verify the differences between two versions of a drawing. After storing a picture of the drawing, you can later retrieve it over the drawing. Anything that is the same cancels out, leaving on the screen only the differences between the two.

To generate a report that lists the differences between two objects, choose **Advanced** from the DRAWING menu and **Integrate** from the ADV DWG OPTS menu.

After you have retrieved an integration project file and the difference report, source object, and target object are displayed, the INTEGRATE menu appears, listing commands for performing the integration:

- **Next**—Move to the next item (difference) in the report.
- **Previous**—Move to the previous item (difference) in the report.
- **Show**—In the active window, highlight the entity (feature, component, and so forth) corresponding to the current item in the difference report. (This command is accessible only if the current item belongs to the object in the current Pro/ENGINEER Window.)
- **Info**—Open the Information window, which contains information about the current item in the difference report (model name, component number, internal ID number, part name, and parents and or children).
- **Action**—Open the ACTION menu. When you specify an action for one occurrence of an item (feature, component, and so forth) in the report, the specified action is set automatically for that item every time it appears in the report.

The ACTION menu contains the following commands:

- **To Merge**—Mark this source item to be merged automatically into the target.
- **To Delete**—Mark this item for automatic deletion from the target.
- **To Ignore**—Ignore this item (that is, take no merge or delete action on it even though a difference is noted).
- **Clear**—Clear all pending actions or instructions to ignore an item.
- **Do Manual**—Integrate the item manually. This involves copying the item manually from the source object to the target object.
- **Merged**—Mark an item as "Merged" after it has been merged manually.
- **Deleted**—Mark an item as "Deleted" after it has been deleted manually.
- **All Changes**—Accept and integrate all changes to the source and target drawings.
- **Source Changes**—Accept and integrate the changes to the source drawing.
- **Target Changes**—Accept and integrate the changes to the target drawing.
- **Unspecified**—Allows the indicated action to be applied to all items that do not already have an action assigned to them.
- **Overwrite**—Allows you to assign an action to all items, even if actions have already been assigned to the items. The new action overwrites the existing action.

- **Current Item**—Allows you to apply an action to the current item.
- **All Items**—Allows you to apply an action to all items that can be acted on.
- **By Type**—Allows you to apply an action to items by type: DIM COSMETICS or PARAMETERS.
- **Execute**—Perform automatically the actions specified in the ACTION column of the difference report to resolve differences between the source and target objects. Each item is marked as Merged, Deleted, or Ignored according to the action that was specified for it.
- **Save Report**—Save the difference report in your local working area as an ASCII text file.
- **Merge View**—Display the difference report with the items sorted in the order in which they have to be integrated to correctly resolve all the external references (for example, parents must be merged before children and children must be deleted before parents).
- **Diff View**—(Default) Display the difference report sorted by object and, within object, by item type. This display gives a quick report of where and what the differences are.

About Hole Charts

By using the ADV DWG OPTS menu in Drawing mode, you can create hole chart tables for drawings. When you create a hole chart table, the system automatically creates a table for drillable hole features in a specified view. The following information is specified in a hole chart:

- Location in X and Y coordinates
- Hole diameter

You can also create tables for datum points and axes. When you create a table listing datum points, their locations are listed in X, Y, and Z coordinates. When you create a table listing datum axes, their locations are listed in X and Y coordinates.

The HOLE TABLE menu presents options that allow you to create a hole chart table, set up your hole chart table, update the hole chart table, or delete a hole chart table:

- **Set Up**—Allows you to set up the format for your hole chart table by:
- **Num Decimals**—Sets the number of decimals in values in the hole chart or table.
- **Param Column**—Defines a parameter column for the hole chart or table. When adding a parameter column, the parameter type must be a feature parameter.
- **Label Size**—Defines the size of text used for labeling holes.
- **Max Rows**—Defines the maximum number of rows in a hole chart or table.
- **Hole Naming**—Defines the method of naming holes (alphanumeric or numeric).
- **Sort Setup**—Modifies the protocol for sorting entries in the hole chart or table.
- **Create**—Allows you to create one of the following:
- **Holes**—Creates a hole chart table.
- **Datum Points**—Creates a table of datum points.
- **Datum Axes**—Creates a table of datum axes.
- **Update**—Updates changes to a hole chart table or table on the current drawing.
- **Delete**—Deletes the entire hole chart table or table.

After you create the hole chart, the holes are labeled on the drawing numerically for ISO standards or alphanumerically for ASME standards.

Note: You cannot update the table any other way other than using the HOLE TABLE menu.

To Create a Hole Chart, Datum Points Table or Datum Axes Table

1. Click **DRAWING > Advanced > ADV DWG OPTS > Hole Table > HOLE TABLE > Create**.
2. From the **List Type** menu, choose one of the following:

- **Holes**—Creates a hole chart table that lists the location of a hole in X and Y coordinates and lists its diameter.
- **Datum Points**—Creates a table that lists the locations of datum points in X, Y, and Z coordinates.
- **Datum Axes**—Creates a table that lists the locations of datum axes in X and Y coordinates.

3. Select the coordinate system for listing.
4. Select the location for the top left corner of the hole chart or table. This action creates the hole chart or table.

Note: Only drillable hole features can be listed in the hole chart.

About Including a Bill of Materials in a Drawing

If you do not have the Pro/REPORT module, but you want to add a Bill of Materials (BOM) to your drawing, you can create a BOM file in Assembly mode by choosing **BOM** from the Pro/ENGINEER **Info** menu. Add the BOM file to the drawing as a note entered from a file. To format or arrange the information in the BOM, you must use the system editor. You can fully edit the BOM displayed on the drawing as a note without affecting the original BOM file.

Note: A BOM that you add to a drawing is not connected with the BOM file that you used to create the note. If the composition of the assembly changes, you must create a new BOM and add it to the drawing as a new note.

To Add a BOM to a Drawing

1. Choose **Insert** > **Note**.
2. Click **Note Types > File** on the Menu Manager.
3. Click **Make Note**. Select a location for the note (BOM) to appear.
4. In the file browser, type the name of the BOM file, including the ".bom" extension. The BOM appears on the drawing.

Note: When adding a BOM to a drawing as a note, justify the note using **Default** or **Left**. If you use **Center** or **Right**, the system might format the BOM incorrectly on the drawing.

About Creating a Snapshot View

The **Snapshot** command in the **VIEWS > Modify View > VIEW MODIFY** menu converts drawing views into a collection of draft entities that are no longer associated with the corresponding model. When you make a view into a snapshot, the following changes occur:

- All visible geometry, axes, datums, and other entities in the view become draft entities.
- All draft entities that were previously associated with the view become free.
- All attached drawing items (such as notes, geometric tolerances, symbols, and draft dimensions) become unattached.
- All visible model dimensions convert to draft dimensions.
- View dimensions become unassociative draft dimensions.

Note: After you create a snapshot, notes and symbols attached to this view remain parametric unless you delete a model using **Del Model**. However, model datums in a snapshot view become draft entities with notes.

Consequences of Creating a Snapshot

After converting a view to a snapshot, the system deletes the view, as well as child views, children on other sheets, and erased children.

To Create a Snapshot of a View

1. Choose **VIEWS** > **Modify View** > **Snapshot**.
2. Specify view(s) by doing one of the following:
   - Choose **All Views** to make snapshots of all views of all models in the drawing. Since draft entities are associated only with the first model of a drawing, the first model must be the active model for you to be

able to manipulate the resulting draft entities.

  – Choose **Pick View**. In a multimodel drawing, this command makes snapshots of views of the current model only.

3. As you select each view, confirm that you would like to convert it to draft entities.
   Note: After you create a snapshot, the display of view arrows might change.

About Using Drawing Overlays

Using overlays, you can superimpose selected views or an entire sheet of one drawing over the current drawing sheet. This functionality is available for drawings, layouts, reports, and diagrams; each can reference objects of these four types, as well as formats.

When working with overlays, keep in mind the following:

• Overlays are *read-only* in the current drawing.

• The system updates overlays to changes in the source drawing.

• Overlaid views appear with all detail items.

• You cannot select overlays for any drawing procedure.

• If the size of the current drawing is different from the size of the source drawing, the overlays brought into the current drawing maintain the same screen size (that is, they occupy the same portion of the graphics window) as they had in the source drawing.

To Create an Overlay

1. Choose DRAWING > **Advanced > Overlay > Add Overlay > Place Views**.
2. Position the overlay properly by doing one of the following:
   – If the source drawing object has only *one* visible view on each of the sheets, position the origin by using the GET POINT menu to select a new location. Press the middle mouse button or choose **Done**.
   – If the source drawing object has *more than one* view on any sheet, choose PLACE VIEWS > **Select Views** and select the desired views. (If the view that you want to use as an overlay is on another sheet, choose **Change Sheet** and select the required sheet). Locate the origin of the main view by selecting a point on the screen. To accept the current position of the view origin, press the middle mouse button or choose GET POINT > **Done**. The system locates all of the dependent views correspondingly.

To Overlay a Drawing onto the Current Drawing

1. Choose DRAWING > **Advanced > Overlay > Add Overlay > Place Sheet**.
2. If the source drawing has only one sheet, the system confirms it and overlays the source drawing onto the current sheet.
   Otherwise, if it has more than one sheet, type the appropriate sheet number. If you do not remember the required sheet number, type [?] to display the source drawing in the subwindow. Find the sheet using commands in the PLACE SHEET menu; then choose **Done**. The specified sheet overlays the current drawing sheet.

To Delete an Overlay

1. Choose DRAWING > **Advanced > Overlay >** OVERLAY DWG > **Del Overlay**.
2. Do one of the following:
   – Choose **Sel Overlay** to specify an overlay to delete by selecting the source drawing object from the SEL OVERLAY menu.
   – Choose **All Overlays** to delete all overlays.

To Move an Overlay

1. Choose DRAWING > **Advanced > Overlay >** OVERLAY DWG > **Move Overlay**.

2. From the SEL OVERLAY menu, select an overlay name or select it on the screen.
3. From the SEL OVLY ITM menu, select a view. Specify the translation vector by using commands in the GET VECTOR menu.
   Note: You can move an overlay by choosing OVERLAY DWG > **Move Overlay**; however, you cannot move *overlaid sheets*.

About Performing a Representation by View

Using the **Represent** command in the VIEWS menu, you can simplify an assembly in a drawing by performing a *representation by view*. When performing a representation by view, keep in mind the following:

- You can use **Simplify** on a given drawing view only once. If you use it a second time on a view, the system restores the previous simplifications when it completes the second simplification.
- Replacing a model in a drawing removes all representations from the drawing.
- You cannot remove from the display any components of a subassembly if assembly constraints are using them.
- You can suppress a feature that you used for view orientation.
- The simplified model in Drawing mode is the *same database object as the original*, and contains no specific information except that pertaining to suppressed features or members.

   Note: To simplify models, you can also use more flexible techniques such as family tables (with a simplified instance that has suppressed features), member display (by blanking components), and simplified representations (by excluding models or features).

To Simplify an Assembly in a Drawing

1. Choose INSTANCES > **Simplify**; then choose **All Views** or **Pick View**. The assembly appears in a small window.
2. Select the component of the assembly to simplify.
3. Choose COMP TYPE > **Rep Part** to simplify the model at the part level by suppressing features; or, choose COMP TYPE > **Rep Subasm** to simplify the model at the subassembly level by suppressing a part.
4. A subwindow appears containing the selected part or subassembly. Choose INSTANCES > **NEW** or type the name of a previously created representation.
   - If the representation is new, select the feature or part to suppress.
   - If you select a detailed view or parent of a detailed view for representation, the system displays a box around both the parent view and the detailed view, indicating that it is going to perform this procedure in both views.

   The system simplifies the top-level assembly accordingly.

About Using Simplified Representations

Using commands in the DWG MODELS menu, you can retrieve, modify, and remove assembly model simplified representations, as well as create assembly views using simplified representations. Pro/ENGINEER limits simplified representations to assembly views, and you *must* specify them before adding a view. You can add multiple views of an assembly, each of a different representation, to a drawing. Once you have added a view, you can change the representation applied in that view only by deleting the view and adding a new view with a new representation.

To Retrieve an Assembly Model Simplified Representation

1. Choose DWG MODELS > **Add Model**.
2. Type an assembly name. The SELECT REP menu (in Assembly mode) appears when you retrieve the initial assembly and for all assembly models that you add subsequently. It lists the Master Representation and all simplified representations that exist for the current assembly. If none have been created for the assembly

model, it does not appear, and the system automatically retrieves the Master Representation.

3. Select a representation. The system applies that representation to subsequent views. You can accept the Master Representation as the default, or select a different representation to retrieve.

Changing the Representation of an Assembly Model

After adding an assembly model to a drawing, you can change its representation by choosing **Set/Add Rep** from the DWG MODELS menu. However, the current model *must* be an assembly model. Selecting a view sets the current representation to the representation shown in that view—you can select *only* views of the current representation. You can use **Sel By Menu** to access a list of all representations that exist for the current assembly. The system makes the specified representation the active representation and applies it to subsequent views. If simplified representations have been made for the assembly model, it automatically retrieves the Master Representation.

When creating simplified representations in Drawing mode and making changes to them, you should keep in mind the following:

- In Assembly mode, simplified representations can contain substitute parts. In Drawing mode, you cannot apply dimensions to these substitute parts.

- When you make changes to a simplified representation in Assembly mode, you may lose information in the drawing. For example, when you change the status of a component by substituting a previously included component, all references to the substituted component are lost.

- When referencing an assembly or its components in a simplified representation, the system can find and use only those components that actually exist in the simplified representation. Specifically, when you orient a view, you cannot use datums belonging to or placed according to components that are not in the current simplified representation.

- You can make projected, auxiliary, revolved, and detailed views from a general view of the same simplified representation.

To Remove Simplified Representations from Session

1. Choose DWG MODELS menu > **Remove Rep**. (The current model must be an assembly model having more than one representation in session.) The RMV REP menu appears, listing only the representations that the drawing references.

2. Select a representation that a view is not using. You can choose one view at a time. The system prompts you for verification before deleting each one.

Removing Simplified Representations from Session

Once you add a representation of an assembly model to a drawing, it remains in session until you explicitly remove it. If a drawing view refers to the particular simplified representation you choose to delete, the system does not allow you to delete it while the drawing is in session, and it attempts to retrieve the Master Representation. If you do remove it from session and delete it, this causes an error when you retrieve the drawing. The system prompts you to select an existing representation from the RMV REP menu, and assigns this representation to all views that previously referred to the deleted representation. If the Master Representation of the assembly no longer exists, you cannot retrieve the drawing.

To Replace a View of a Simplified Representation

You can replace a view of a simplified representation (including the Master Representation) with another simplified representation.

1. Choose VIEWS > **Modify Views > View State**.
2. Select a drawing view to replace.
3. Select a simplified representation from the list of available representations to use as a replacement.

About Using Geometry Representations

When working with simplified representations in Drawing mode, you can use *geometry representations*—a type of advanced simplified representation. Geometry representations require less time to retrieve than the actual part because the system does not retrieve any of the parametric information, only the geometry. You can use them to remove hidden lines, obtain measure information, and accurately calculate mass properties.

When working with geometry representations, keep in mind the following:

- By default, you cannot create drawing references to geometry representations (this includes dimensions, notes, and leaders), but if you set the configuration file option `allow_reps_to_geom_reps_in_drws` to `yes`, you can create references. However, these references may become invalid if the referenced geometry changes. This option is for advanced users who are aware that some references to geometry representations may not be updated in drawings.
- Graphics representations are not available. If you include graphics representations in a drawing view of a simplified representation, the system changes those representations to geometry representations. For more information, refer to the *Assembly Modeling User's Guide*.

About Creating Drawing Programs

Using the **Program** command in the ADV DWG OPTS menu, you can create a *drawing program* to define how a drawing will adapt to a change in the state of its model. Changes might occur when you re-execute the model with new Pro/PROGRAM inputs, or replace it with another instance from a family of models. Typically, you can distinguish one sequence, or state, from another by the values of some of the parameters in the model.

A drawing program is intended to be used to adapt a drawing to a part or assembly program. It contains logic statements that control the drawing layout and perform various detail functions. For example, a part drawing might include a detailed view of a particular feature, such as a keyway. If you suppress that feature, the system should erase that detailed view, and move the other views to fill the space. If you make a state such as `no_detail_view` that erases the detailed view and organizes (moves) the other views appropriately, the program queries the system to determine if the keyway is suppressed. If so, the drawing displays the `no_detail_view` state. If not, the drawing displays the detailed views of the model.

A drawing program has two portions: *states* and *program text*. A state is a named sequence of familiar procedures, such as showing a dimension or moving a view, that you perform on a drawing to define how it should appear. It is a record of modifications that you would like to make to the drawing. To create a state, you type a name and then record various detail commands. You can *play back* these commands to determine what the drawing state actually does, and then edit it, if necessary. As you create a drawing, you can create additional states and delete others.

You can place dimensions, dimension breaks, and dimension clips on snap lines during the creation of a drawing program state.

Once you have defined the drawing states, you can create a drawing program, as shown in the preceding figure. The drawing program is a text file, embedded inside the drawing, which contains lines of text that set certain states for the drawing, depending on the values of the conditional expressions that you use. You can use IF statements, drawing parameters, and assignment statements to set previously defined drawing states. The program first searches the drawing for a drawing parameter; if it does not find a parameter, it searches the default model for a model parameter. If the drawing program recognizes the drawing parameter, it designates it with the postfix ":d" (that is, drawing_attribute:d). If it is a model parameter, it designates it with the model number.

- Commands to execute a drawing state take the following form: SET STATE *name_of_state.*
- You can nest IF statements in the following form:

```
IF <expression>
    ELSE IF <expression>
    ELSE
    ENDIF.
```

An expression is a logical expression that you could use in a part relation. It could contain drawing attributes (as found in the SET UP menu) or the feature suppressed function (that is, FEAT_SUPPRESSED (model_name, feat_id) to determine if a feature is suppressed; and FEAT_SUPPRESSED (as*sembly_name, comp_id*) to determine if a component is suppressed). For example:

```
IF FEAT_SUPPRESSED (bolt,15)
SET STATE no_detail_view
```

In this case, the system checks to see if feature ID 15 of the model named "bolt" is suppressed. If so, it sets the state to `no_detail_view`; otherwise, it does nothing.

- Assignment statements take the following form: *var_name = expression,* where *var_name* is a variable name

- Comment lines take the following form: /* *This is a comment line.*

  Note: The words IF, ELSE, ENDIF, SET, STATE, FEAT_SUPPRESSED are reserved. You cannot use them as variable names in a drawing program. Also, if using the equal sign (=) in an IF statement, use "==". If using a statement to set a parameter value, use "=".

Example: Drawing Program Text

IF FEAT _SUPPRESSED (shaft, 12)
SET STATE no_detail_view
ELSE SET STATE all_detail

To Create a Record of Modifications to a Drawing (a State)

1. Choose DRAWING **> Advanced > Program > Define States > Create State**.
2. Type the name of the state.
3. Choose **Record Cmds**.
4. From the DWG COMMANDS menu, choose commands to move, erase, and show items in the drawing; then save these changes (later, you can edit the state and highlight those items).
5. To undo any of the modifications that you have made, choose EDIT STATE > **Undo Cmds**. As shown in the illustration of the Undo State Commands text-tool window, the system places the cursor on the command that you last executed, and highlights in the drawing window the item that was acted on. If it highlights a command line in red, it means that the system has *undone* it (the *name* field is not always filled in because not all items have names).
6. After you create the state, the system resets the drawing to its default display. Choose **Done** to activate the state and make the changes.

The EDIT STATE menu displays the following commands:

- **Record Cmds**—Displays the DWG COMMANDS menu, a compound menu of possible commands and object types upon which to operate. After choosing an action and item type, you can modify items in the drawing. The following table presents those commands that you can record (marked with an "X"). In addition to those commands included in the table as not being recordable, you cannot record the following: moving leader jogs or dimension jogs; modifying the model grid; erasing, showing, and moving cross-section arrows and view notes; erasing or moving ordinate dimensions; and modifying filled draft cross sections.

- **Play Cmds**—Resets the display of the drawing to its original display (before starting the state), and plays each saved command in the state, pausing in between. You can stop playback by selecting the stop sign icon in the message area. The system skips commands that do not affect any displayed sheet.

- **Undo Cmds**—Removes commands. Displays the Undo State Commands text-tool window listing commands in the state.

- **Layer States**—Displays the DPM LAYER menu. You can blank, display, or set to normal the display status of any drawing layer. You can also leave the status unchanged. Drawing programs can *only* control the status of drawing layers and cannot affect model layers.
- **Set Cur Sheet**—Switches to another sheet of the drawing to modify it. You can also display another drawing sheet by choosing **Windows** > **New** from the Pro/ENGINEER menu bar. This command does *not* affect the current state in any way.

Commands That You Can Record (marked with an "X")

DETAIL ITEMS	Move	Mod Attach	Erase	Show	Show View	Switch Sheet	Create
Dimensions	X	X	X	X	X		X
Reference Dimensions	X	X	X	X	X		X
Notes	X	X	X	X	X	X	X
Balloon Notes	X	X	X	X	X	X	X
BOM	X	X	X	X	X		
Balloons							
Symbols	X	X	X	X	X	X	
Gtols	X	X	X	X	X	X	
Surf Finish Symbols	X	X	X	X	X	X	
Datum	X		X	X			
Targets							
Set Datums	X		X	X			
Axes	X		X	X			
Draft Entities	X					X	
Draft Datums	X					X	
Draft Axes	X					X	
Draft Groups	X					X	

Tables	X				X
Views	X		X	X	X

The UNDO COMMANDS menu displays these commands:

- **Next**—Executes the command at the next line and moves the cursor to that line.
- **Prev**—Unexecutes the command at the previous line and moves the cursor to it.
- **Run**—Selects a line in the text-tool window. The system executes or unexecutes commands until that line.
- **Toggle Cur**—Undoes the command at the current line. The system updates the drawing window to this, and highlights the command line. If you have already undone the current command, this command restores it.
- **Pick/Toggle**—Selects a command line in the text-tool window. If you choose a command that you have not undone, it undoes it. If you choose a command that you have undone, it restores it.

To Create Detail Items in a Drawing State

To create detail items such as dimensions, notes, and balloon notes in a drawing state, choose **Create** from the DWG COMMANDS menu. Items that you create in one state are not visible in any other state or outside of the drawing program.

To Redefine a Drawing State

To *redefine a state*, choose **Edit State** from the DRAW STATES menu. From the PRGM STATES menu, choose the state to edit. If the drawing program is not using the state, the system plays it quickly; otherwise, it rolls the program to the point where this state is executed, and displays the EDIT STATE menu.

To Remove a Drawing State

To *remove a state*, choose **Delete State** from the DRAW STATES menu. You can delete any state that the drawing program is not currently using.

To Call a User-Defined Function

To call in a user-defined function that is already registered in a Pro/TOOLKIT application, choose **User Func** from the DWG COMMANDS menu.

The Edit Program Menu

To create a drawing program, choose **Edit Program** from the DRAW PROGRAM menu. The EDIT PROGRAM menu displays the following commands:

- **Insert Line**—Prompts you to type text lines into the program. To finish, type a carriage return.
- **Delete Line**—Deletes a range of lines from the program.
- **Edit Line**—Modifies the current line in the message area.
- **Screen Edit**—Edits the program directly in the text-tool window.
- **File Edit**—Edits the drawing program using the system editor.
- **Set State**—Displays a run-time menu of all drawing states. You can select a state from the menu, or create a new state interactively.
- **Next**—Moves to the next executable line of the program.
- **Prev**—Backs up one line, undoing the modification that line caused to the drawing. If you get to this line by using the **Jump** command, you back up to the line on which the jump was started.

- **Run**—Executes the program forward or backward until reaching a certain line.
- **Jump**—Sets the current line as a specified line number without executing any line. You can only jump to executable lines.
- **Switch Dim**—Changes the data display to symbols or numerical values. This command does not change the program or the drawing.
- **Set Cur Sheet**—Sets the current sheet of the drawing for display purposes. You can display another drawing sheet by choosing **Windows** from the Pro/ENGINEER menu bar, followed by **New**. This command does not change the program or the drawing.

To Run the Drawing Program (Execute a State)

Along with the EDIT PROGRAM menu, the DRAW PROGRAM text-tool window opens, listing the lines of the program. Using the commands in the EDIT PROGRAM menu, you can interactively add lines to the program, remove lines from it, step forward and backward, and modify it. As you make each change, the system updates the display of the drawing.

> **Notes:**
> - You cannot modify a value that the program is driving (such as the value of a drawing user attribute, the sheet on which a draft entity should be). However, you can modify a value that the program is not driving if the program moves the view.
> - If you delete an item that the drawing program controls, the system automatically removes from the state the command in the program that controls that item.
> - If you convert a view to a snapshot, the system removes from their states all commands in the drawing program that control the view and its subordinate items.
> - The drawing program can only modify the drawing, not the models of the drawing.

When the program appears, the system positions the text-tool cursor after the last line executed. As you move through the program, you can change the display by choosing commands from the SET STATE menu. After you exit the DRAW PROGRAM menu, the system executes the program in its entirety and redisplays the drawing. It automatically updates the drawing program to reflect changes made to parts when you retrieve a drawing, switch to Drawing mode, or regenerate by choosing **Draft** from the REGENERATE menu (you can use **Model** *only* if the model changes).

After you use one of the editing commands, the system reinterprets the entire text of the program, and re-executes the program up to the cursor.

- If the interpreting is successful, it converts all parameter names, model names, and keywords to uppercase. In addition, the model postfix of a parameter appears automatically when needed (for example, if the drawing has more than one model).
- If the interpreting is unsuccessful, the system informs you that an error has occurred and highlights the error line. It cannot execute any line that is not interpreted, so if you move the text-tool cursor into an uninterpreted area of the program, it does not execute any states and does not change the drawing. Until you correct the error, the system does not execute the program and make the changes that the program has made to the drawing.

About Setting the Size of a Drawing View

You can set the size of a drawing view so that when you change dimensions of the model, the size of the drawing view stays constant relative to a given model dimension. To do this, specify the model parameter `drawing_scale_factor`, used in Drawing mode as a view scale factor. This parameter is a scale factor that the drawing scale multiplies to determine the actual size of the entire drawing view.

Note: This does not change the value of "d#" in the drawing or the overall drawing scale, only the spatial dimension of the displayed entities.

The following table shows how the system calculates the size of the drawing view using the drawing scale and the `drawing_scale_factor` model parameter.

Calculating the Size of a Drawing View

value of d#	Length para-meter	Drawing _scale _factor	drawing scale	Length in the View	Comments
5	5	1	2	10	*Changing drawing scale*. Drawing scale changed to 2; the edge length in the view becomes 10.
5	5	1	1	5	***Initial conditions.*** Model edge has a corresponding dimension d# = 5. If the drawing scale is 1 and the edge length is 5, the "drawing_scale_factor" is set to 1.
10	5	.5	1	5	***Changing dimension value.*** Value of d# changed to 10 in Part or Assembly mode, so "drawing_scale _factor" becomes .5. Makes the calculated length in the view half of its dimension value (it remains 5 (unchanged)).
5	10	2	1	10	***Changing value of "length" parameter.*** Value of "length" parameter changed in the relation to 10, so "drawing_scale_factor" becomes 2. Multiplies the length of the edges in the view by 2.
5	10	2	2	20	***Changing value of "length" parameter at a given scale.*** The edge with the dimension value of 5 appears in the drawing with a scale of 2. Changing "length" parameter to 10 changes "drawing_scale_factor" to 2, so the calculated length of the edge in the drawing is 20.

To Set the Size of the Drawing View

1. Determine the edge that you want to use as the basis for setting the `drawing_scale_factor`. The system uses the dimension value and the length of this edge in the drawing to calculate it. Whenever you change either one of these variables ("length" or d#), it calculates the length of model edges according to the following equation:
   ```
   length in the view = (dimension value) x(drawing_scale_factor) x (drawing_scale)
   ```
2. Add the following relation to the part or assembly:
   ```
   drawing_scale_factor = length / dimension
   ```
   where length is the parameter corresponding to the current drawing measurement of the above dimension d#, and dimension is a model dimension, d#.
3. Regenerate the model.

Pro/DETAIL

About Setting Draft Scale

By setting a draft scale using the drawing setup file option `draft_scale`, you can create draft entities with a scaled size. If you set it at 0.5, any entities that you create subsequently are one-half the size that you specified. For example, creating a horizontal line of 4 inches (by selecting one endpoint and placing the second endpoint using **Rel Coords** with x=4 and y=0), the line measures as 2 inches on paper, but dimensions as 4 inches. If you change the draft scale after creating and associatively dimensioning draft entities, the dimensions change.

The draft scale, like other parameter values in the drawing setup file, is completely independent of a drawing scale. When the drawing size changes, the value of the draft scale does not change. The system scales the draft geometry by the proportion necessary to maintain its relative location on the sheet, and it appears to be the same size that it was before. The geometry is, in fact, a different size (because the sheet is a different size), and this is reflected by any new dimensions that you create for it; the ratio of the new dimension to the previous one is equal to the ratio of the size of the new sheet to the old one. If you set the drawing setup file option `associative_dimensioning` to `yes` (the default), when the size of dimensioned draft entities changes, the system updates the dimensions to the latest value).

Example: Drawing Setup File

h_dr_det_setup_create_ex.doc

```
!
! These options control text not subject to other options
!
drawing_text_height                 0.156250
text_thickness                      0.000000
text_width_factor                   0.800000
!
! These options control views and their annotations
!
broken_view_offset                  1.000000
create_area_unfold_segmented        YES
def_view_text_height                0.000000
def_view_text_thickness             0.000000
detail_circle_line_style            SOLIDFONT
detail_view_circle                  ON
half_view_line                      SOLID
projection_type                     THIRD_ANGLE
show_total_unfold_seam              YES
view_note                           STD_ANSI
view_scale_denominator              0
view_scale_format                   DECIMAL
!
! These options control cross sections and their arrows
!
crossec_arrow_length                0.187500
crossec_arrow_style                 TAIL_ONLINE
crossec_arrow_width                 0.062500
crossec_text_place                  AFTER_HEAD
cutting_line                        STD_ANSI
cutting_line_adapt                  NO
```

cutting_line_segment	0.000000
draw_cosms_in_area_xsec	NO
remove_cosms_from_xsecs	TOTAL
!	
! These options control solids shown in views	
!	
datum_point_size	0.312500
datum_point_shape	CROSS
hlr_for_pipe_solid_cl	NO
hlr_for_threads	YES
location_radius	DEFAULT(2.)
mesh_surface_lines	ON
thread_standard	STD_ANSI
hidden_tangent_edges	DEFAULT
ref_des_display	NO
!	
! These options control dimensions	
!	
allow_3d_dimensions	NO
angdim_text_orientation	HORIZONTAL
associative_dimensioning	YES
blank_zero_tolerance	NO
chamfer_45deg_leader_style	STD_ASME_ANSI
clip_dimensions	YES
clip_dim_arrow_style	DOUBLE_ARROW
default_dim_elbows	YES
dim_fraction_format	DEFAULT
dim_leader_length	0.500000
dim_text_gap	0.500000
draft_scale	1.000000
draw_ang_units	ANG_DEG
draw_ang_unit_trail_zeros	YES
dual_digits_diff	-1
dual_dimension_brackets	YES
dual_dimensioning	NO
dual_secondary_units	MM
iso_ordinate_delta	NO
lead_trail_zeros	STD_DEFAULT
lead_trail_zeros_scope	DIMS
ord_dim_standard	STD_ANSI
orddim_text_orientation	PARALLEL
parallel_dim_placement	ABOVE
shrinkage_value_display	PERCENT_SHRINK
text_orientation	HORIZONTAL
tol_display	NO
tol_text_height_factor	STANDARD
tol_text_width_factor	STANDARD
use_major_units	NO

witness_line_delta	0.125000
witness_line_offset	0.062500
!	
! These options control text and line fonts	
!	
default_font	font
aux_font	1 filled
!	
! These options control leaders	
!	
draw_arrow_length	0.187500
draw_arrow_style	CLOSED
dim_dot_box_style	DEFAULT
draw_arrow_width	0.062500
draw_attach_sym_height	DEFAULT
draw_attach_sym_width	DEFAULT
draw_dot_diameter	DEFAULT
leader_elbow_length	0.250000
!	
! These options control axes	
!	
axis_interior_clipping	NO
axis_line_offset	0.100000
circle_axis_offset	0.100000
radial_pattern_axis_circle	NO
!	
! These options control geometric tolerancing information	
!	
gtol_datums	STD_ANSI
gtol_dim_placement	ON_BOTTOM
new_iso_set_datums	YES
asme_dtm_on_dia_dim_gtol	ON_GTOL
!	
! These options control tables, repeat regions, and BOM balloons	
!	
dash_supp_dims_in_region	YES
def_bom_balloon_leader_sym	ARROWHEAD
model_digits_in_region	YES
show_cbl_term_in_region	YES
2d_region_columns_fit_text	NO
!	
! These options control layers	
!	
draw_layer_overrides_model	NO
ignore_model_layer_status	YES
!	
! These options control model grids	
!	

```
model_grid_balloon_size                         0.200000
model_grid_neg_prefix                           -
model_grid_num_dig_display                      0
model_grid_offset                               DEFAULT
!
! These options control theoretical piping bend intersection
!
show_pipe_theor_cl_pts                          BEND_CL
pipe_pt_shape                                   CROSS
pipe_pt_size                                    DEFAULT
!
! Miscellaneous options
!
decimal_marker                                  COMMA_FOR_METRIC_DUAL
drawing_units                                   INCH
line_style_standard                             STD_ANSI
max_balloon_radius                              0.000000
min_balloon_radius                              0.000000
node_radius                                     DEFAULT
sym_flip_rotated_text                           NO
weld_symbol_standard                            STD_ANSI
yes_no_parameter_display                        TRUE_FALSE
default_pipe_bend_note                          NO
```

2d_region_columns_fit_text

no, yes

Determines whether each column in a two-dimensional repeat region is autosized to fit the longest piece of text in each column.

If set to "yes," autosizes each column in 2-D repeat regions so that the longest text in the column fits within the column and does not overlap adjacent columns or force large gaps in the table. Columns with no text in them use the default column width for the region (the width of the template cell).

Columns of tables that contain autosized 2-D repeat regions cannot be manually resized.

allow_3D_dimensions

no, yes

Determines if dimensions are shown in isometric views.

angdim_text_orientation

horizontal, parallel_outside, horizontal_outside, parallel_above, parallel_fully_outside

Controls the placement of angular dimensions in drawings. If set to "horizontal," displays text of angular dimensions horizontally at all times, centered between the leaders (equivalent to the value "horizontal" for the drawing setup file option "text_orientation"). If set to "parallel_outside," displays text parallel to the leader lines, regardless of their orientation (equivalent to the value "parallel" for the drawing setup file option "text_orientation"). If set to "horizontal_outside," displays text horizontally outside the dimension. If set to "parallel_above," displays text parallel to the dimension arc, but above it. If set to "parallel_fully_outside," displays text of angular dimensions (with a plus/minus tolerance) parallel to the leader lines. For more information and an illustration, see How to Display Dimension Text Symbols on page 6 - 38.

asme_dtm_on_dia_dim_gtol

on_dim, on_gtol

Controls the placement of a set datum attached to a diameter dimension. If set to "on_dim", the set datum attaches to the diameter dimension. If set to "on_gtol", places it on the gtol in accordance with the ASME standard

associative_dimensioning

yes, no

Associates draft dimensions to draft entities. The system associates only dimensions that you create while you have this set to "yes" (see Draft Dimensions on page 6 - 5).

aux_font

1 filled, # font index name (# = 1 through 8)

Sets the auxiliary text font '#' as the font identified in the specified font index. The system remembers the number of the auxiliary font as corresponding to the name of the font that appears in the TEXT FONTS menu listing available fonts. Do not include the suffix ".ndx"; for example, "1 gothicfont."

PTC and TrueType fonts are available. See default font HELP topic for detailed information.

aux_line_font

font name (#=1 through 10,000)

Sets auxiliary line fonts as the font specified. Uses the integer number to associate a line font to draft geometry. In this way, you can effect a blanket change by changing the font name associated with a number. *You must add this option to the drawing setup file whenever you want to set the auxiliary font.*

axis_interior_clipping

no, yes

If set to "no," displays axes in a drawing according to the ANSI Y14.2M standard.

If set to "yes," you can adjust each axis individually by clipping and moving.

axis_line_offset

0.100000, value

Sets the default distance that a linear axis extends beyond its associated feature.

blank_zero_tolerance

no, yes

Controls the display of a plus or minus tolerance value. If set to "yes," does not display a plus or minus tolerance value if you set the tolerance value to zero.

broken_view_offset

1.000000, value

Sets the offset distance between the two halves of a broken view.
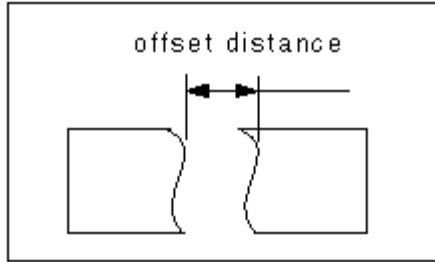
offset distance

chamfer_45deg_leader_style

std_asme_ansi, std_din, std_iso, std_jis

Controls the leader type of a chamfer dimension without affecting the text.

circle_axis_offset

0.100000, value

Sets the default distance that a circular cross-hair axis extends beyond the circular edge.

clip_diam_dimensions

yes, no
Automatically clips the diameter dimensions at the view border. Dimension endpoints outside of the view border are clipped to the view border. No clipping occurs when both endpoints are inside view border. The default is *no* for drawings created in earlier releases.

clip_dim_arrow_style

double_arrow, arrowhead, dot, filled_dot, arrow, slash, integral, box, filled_box, none

Controls the arrow style of clipped dimensions.

clip_dimensions

yes, no

Controls the display of dimensions in a detailed view. If set to "yes," does not display dimensions completely outside of a detailed view boundary; shows dimensions that cross a detailed boundary with a special double arrow. If set to "no," displays all dimensions.

create_area_unfold_segmented

yes, no

Makes the display of dimensions in area unfolded cross-sectional views similar to those in total unfolded cross-sectional views. If set to "yes," displays the view in segments when creating a new view—one piece at a time—corresponding to the straight segments of the cross-sectional sketch. [To draw view borders between view segments, set "show_total_unfold_seam" to "yes."] This option only affects new views. Pro/ENGINEER does not support detailed views of segmented area unfolded cross-sectional views or total unfolded cross-sectional views. However, it does support detailed views of non-segmented area unfolded cross-sectional views.

crossec_arrow_length

0.187500, value

Sets the length of the cross-section cutting plane arrowheads.

crossec_arrow_style

tail_online, head_online

Sets the display style of cross-section arrows.
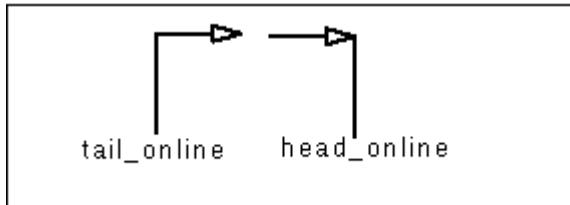
crossec_arrow_width

0.062500, value

Sets the width of cross-section cutting plane arrows.
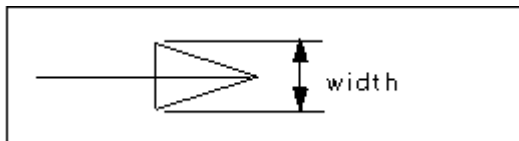
crossec_text_place

after_head, before_tail, above_tail, above_line, no_text

Sets the location of cross-section text. If set to "no text," does not display any cross-section text.
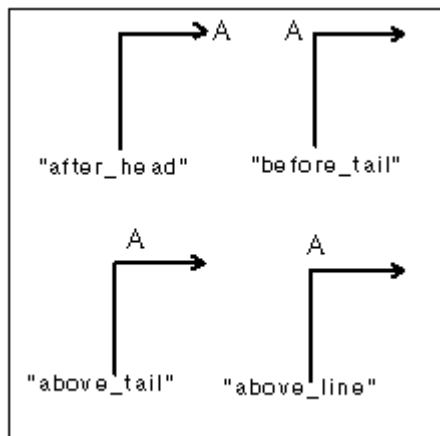
crossec_type

old_style, new_style
Improves the ability to create complicated cross section views in drawings, and reduces or eliminates the number of occurrences when a cross section view cannot be created.

If set to `old_style`, the system uses a cut to remove geometry to create the cross section view. If set to `new_style`, the system uses a z-clipping plane to create the cross section view.

cutting_line

std_ansi, std_din, std_iso, std_jis, std_ansi_dashed, std_jis_alternate

Controls display of cutting line. If set to "std_ansi," uses the ANSI standard for cutting lines. If set to "std_ansi_dashed," uses dashed lines. Otherwise, uses the DIN standard cutting line. Displays its thickened portion in white, and displays its thin portion in gray. If set to "std_jis_alternate" and the drawing setup file option "cutting _line_segment" is set, displays view arrows as follows:

- The thickened portion of the cutting line forms an angle and the system displays it in blue.
- Displays the connecting portions of the cutting line segment between thickened segments in yellow.
- Displays arrow portions in white.

If you set the drawing setup file option "cutting_line_segment" to "0," displays the entire cutting line as a dashed yellow line. If the length of a cutting line segment is too large, the entire cutting line displays in blue.

cutting_line_adapt

no, yes

Controls display of line fonts used to show cross-sectional arrows. If set to "yes," all line fonts display adaptively, beginning in the middle of a complete line segment and ending in the middle of a complete line segment.

cutting_line_segment

0.000000, value

Specifies the length in drawing units of the thickened portion of a non-ANSI cutting line. If set to "0," the length of the cutting line segment is 0.

dash_supp_dims_in_region

no, yes

Controls display of dimension values in Pro/REPORT table repeat regions. If set to "no," displays the values in Pro/REPORT table repeat regions. If set to "yes," suppresses the dimension and displays a dash instead.

datum_point_size

default, value

Controls size of model datum points and sketched two-dimensional points. The system does not use the units of the drawing or model; it always displays the point size in inches.

datum_point_shape

cross, dot, circle, triangle, square

Controls display of datum points.

decimal_marker

comma_for_metric_dual, period, comma

Determines which character marks the decimal point in secondary dimensions.

default_dim_elbows

yes, no

Controls display of dimension elbows. If set to "yes," dimensions display with elbows.

default_font configuration file option

font, font index name

Sets default text fonts as those fonts listed in the specified font index. Do *not* include the ".ndx" extension. The fonts "font" and "filled" are in the setup file.

default_pipe_bend_note

no, value

Controls display of pipe bend notes in drawings. If set as text within quotation marks, uses that value when creating bend notes. Text may include parameters such as "&bend_name:att_pipe_bend" and "&bend_tol: att_pipe_bend". If set as a directory path, references a previously created note saved as a file.

def_bom_balloon_leader_sym

arrowhead, dot, filled_dot, no_arrow, slash, integral, box, filled_box

Sets the default arrow (attach point) style for BOM balloons in reports.

def_view_text_height

0.000000, value

Sets the height of text in view names used in view notes and in arrows in cross-sectional and projection detail views.

def_view_text_thickness

0.000000, value

Sets default thickness for new text in view names used in view notes and in arrows in newly created cross-sectional and projection detail views.

detail_circle_line_style

SOLID FONT, any available system-defined or user-defined line font

Sets line font for circles indicating a detailed view in a drawing.

detail_circle_view_note

Yes No
View note within detail circle

detail_note_text

default, value

Sets the text in the detail view reference note.

When set to `default` (the default setting), detail view reference notes have the localization of `SEE DETAIL<viewname>` for their text.

If set to a different value, detail view reference notes have the localization of `<detail_note_text> <viewname>` for their text.

detail_view_circle

on, off

Sets display of a circle drawn about the portion of a model that is detailed by a detailed view.

dim_dot_box_style

default, filled, hollow

Controls the arrow style display of dots and boxes only for leaders of linear dimensions. When set to "default" (the default setting), uses the "draw_arrow_style" setting.

When set to "filled," fills dots and boxes for arrows of linear dimensions. Use "filled" to have new drawings appear with dots and boxes filled for dimension arrows.

When set to "hollow," dots and boxes for arrows of linear dimensions are not filled.

When set to "default" for BOM balloon arrows, the drawing setup file option "def_bom_balloon_leader_sym" defines the default arrow (attach point) style for BOM balloons in reports.

dim_fraction_format configuration file option

default, std, aisc

Controls the display of fractional dimensions in drawings. Unless set to "default," this option supersedes the configuration file option `dim_fraction_format`.

If set to "std," displays fractional dimensions in drawings in the standard Pro/ENGINEER format.

If set to "aisc," displays fractional dimensions in drawings in the AISC format.

If set to "default" (the default setting), displays fractional dimensions in drawings according to the setting of the configuration file option `dim_fraction_format`.

Set the option to "AISC" to display fractional dimensions in the AISC format. The "AISC" setting also displays architectural units according to AISC format for feet-inches dimensions.

Set the option to "default" to display fractional dimensions according to the setting of the config.pro option "dim_fraction_format".

The drawing setup file option "use_major_units" is used to control whether fractional dimensions show in feet-inches. Use the option as follows:

Set the option to "no" and fractional dimensions are not displayed in major units.

Set the option to "default" to display fractional dimensions according the setting of the config.pro option "use_major_units"

The drawing setup options "dim_fraction_format" and "use_major_units" control the display of dimensions when you retrieve drawings created prior to Release 2000*i*. You can create dimensions and the configuration file option controls the formatting of the fractional dimensions.

The drawing setup option "dim_fraction_format" controls the display of fractional dimensions in drawings. The drawing setup option supersedes the configuration file option "dim_fraction_format" unless the drawing setup option is set to "default."

The drawing setup option "use_major_units" controls the display of fractional dimensions in drawings in the same way that the configuration file option "use_major_units" controls display of fractional dimensions in Part and Assembly modes. The drawing setup option "use_major_units" supersedes the configuration file option "use_major_units" in drawings unless the drawing option is set to "default."

dim_leader_length

0.500000, value

Sets length of dimension leader line when leader arrows are outside of witness lines.

dim_text_gap

0.500000, factor

Controls distance between dimension text and dimension leader line and represents the ratio between gap size and text height.

For the diameter dimension, if "text_orientation" is set to "parallel_diam_horiz," "dim_text_gap" controls the extension of an elbow line beyond the text.

draft_scale

1.000000, value

Determines value of draft dimensions relative to actual length of draft entity on drawing.

draw_ang_units

ang_deg, ang_min, ang_sec

Sets display of angular dimensions in a drawing. If set to "ang_deg," creates decimal degrees; if set to "ang_min," creates degrees and decimal minutes; and if set to "ang_sec," creates degrees, minutes, and decimal seconds.

draw_ang_unit_trail_zeros

yes, no

Controls display of angular dimensions. If set to "yes," removes trailing zeros (in adherence to ANSI standards) when showing angular dimensions in degrees/minutes/seconds format. If set to "no," does not display trailing zeros in angular dimensions or tolerances.

draw_arrow_length

0.187500, value

Sets length of leader line arrow heads.

draw_arrow_style

closed, open, filled

Controls arrow style for all detail items involving arrows, including leaders of dimensions, notes, 3-D notes, geometric tolerances, symbols, and balloons.

draw_arrow_width

0.062500, value

Sets width of leader line arrow heads. Drives these drawing setup file options:

draw_attach_sym_height

draw_attach_sym_width

draw_dot_diameter

draw_attach_sym_height

default, value

Sets height of leader line slashes, integral signs, and boxes. If set to "default," uses value set for "draw_arrow_width."

draw_attach_sym_width

default, value

Sets width of leader line slashes, integral signs, and boxes. If set to "default," uses value set for "draw_arrow_width."

draw_cosms_in_area_xsec

no, yes

Controls display of cosmetic sketches and datum curve features that lie in the cutting plane in planar area cross-sectional views. If set to "yes," shows all cosmetic sketches and datum curve features that lie in the cutting plane. If set to "no," does not show them.

draw_dot_diameter

default, value

Sets diameter of leader line dots. If set to "default," uses value set for "draw_arrow_width."

draw_layer_overrides_model

no, yes

Directs drawing layer display setting to determine the setting of drawing model layers with the same name. If set to "yes," implicitly includes drawing model layers in drawing layers with the same name for purposes of setting the display. If set to "no," ignores nondrawing layers when the display status of layers is set in the drawing model.

drawing_text_height

0.156250, value

Sets default text height for all text in the drawing using value set for "drawing_units."

drawing_units

inch, foot, mm, cm, m

Sets units for all drawing parameters.

dual_digits_diff

-1, value

Controls number of digits to the right of the decimal that the secondary dimension differs from primary dimension. For example, using the default value -1 results in the following when primary units are inches and secondary units are millimeters: 10.235 [259.96].

dual_dimension_brackets

yes, no

Controls display of brackets with dimension units. This option works only when you are using "dual_dimensioning." If set to "yes," displays dimension units that occur second in brackets; if set to "no," does not display brackets.

dual_dimensioning

no, primary [secondary], secondary [primary], secondary

Controls format of dimension display. If set to "no," displays a single value for dimensions. If set to "primary [secondary]," displays dimensions with primary units (established by the model) and secondary units; if set to "secondary," only displays the secondary dimensions of the drawing, as if they were primary.

dual_secondary_units

mm, inch, foot, cm, m

Sets units for the display of secondary dimensions.

gtol_datums

std_ansi, std_ansi_mm, std_iso, std_jis, std_din, std_iso_jis, std_ansi_dashed, std_asme

Sets drafting standard followed for displaying reference datums in drawings. Display affects both axes and datum planes, and display of reference part datums.

std_ansi   std_iso_jis
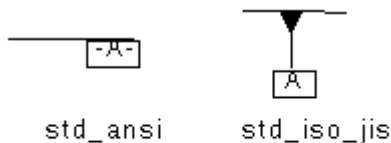
gtol_dim_placement configuration file option

on_bottom, under_value

Determines location of a feature control frame of a geometric tolerance when attached to a dimension symbol that contains additional text. If set to "on_bottom," places the geometric tolerance at the bottom of the dimension symbol, beneath any additional lines of text. If set to "under_value," places the geometric tolerance immediately below the dimension value and before any additional lines of text.

half_view_line

solid, symmetry, symmetry_iso, symmetry_asme, none

 If set to "solid," draws solid lines where material is present. If set to "symmetry," draws a centerline extending beyond the part and acting as a break line. If set to "symmetry_iso," displays a half view symmetry line according to ISO standard 128:1982 5.5— half view symmetry lines are displayed in yellow with thin linestyle, and the "hash" marks at the ends of the symmetry line are also yellow with thin linestyle. If set to "symmetry_asme," displays a half view symmetry line according to ASME standard ASME Y14.2M-1992— half view symmetry lines are displayed in yellow with thin linestyle, and the "hash" marks are displayed in white with thick linestyle.

If set to "none," draws the object a small distance past the symmetry line. You must select an offset datum to create the half view; make sure there is a centerline indicating the actual half.

hidden_tangent_edges

default, dimmed, erased

Controls display of hidden tangent edges in drawing views. If set to "dimmed," plots hidden tangent edges in a view using Pen 7. Lines appear dashed in same color as dimmed visible tangent edges. However, you must select **Hidden Line** or **No Hidden Line** from the **Display Style** list in the Pro/ENGINEER Environment dialog box. If set to "erased," removes all hidden tangent edges automatically from screen and plot.

hlr_for_pipe_solid_cl

no, yes

Controls display of pipe centerlines. If set to "yes," hidden line removal affects pipe centerlines. If set to "no," it does not. Operates only on pipes created in Pro/PIPING, not on pipe features in a part.

hlr_for_threads

no, yes

Controls display of threads in a drawing depending on whether it complies with the ISO or ANSI standard (set by the "thread_standard" option). If set to "yes," thread edges meet ANSI or ISO standard for **Hidden Line** display.

ignore_model_layer_status

yes, no

Controls whether the system considers layer status in models. If set to "yes," ignores changes to all layer status in the models of the drawing made in another mode.

iso_ordinate_delta

no, yes

Improves display of offset between an ISO-ordinate dimension line and witness line, referred to as the "witness line delta." If set to "yes," displays offset correctly, according to value specified for the drawing setup file option "witness_line_delta." If set to "no," does not display offset exactly in accordance with the specified value (it is "off" by about 2 millimeters).

leader_elbow_length

0.250000, value

Determines length of leader elbow (the horizontal leg attached to text).

lead_trail_zeros

std_default, std_metric, std_english, both

Controls display of leading and trail zeros in dimensions. Can also control display of leading and trail zeros in parameters, depending on the setting of the drawing setup file option "lead_trail_zeros_scope." If "lead_trail_zeros_scope" is set to "all," controls the display of leading and trail zeros for dimensions and also for all floating point parameters on a drawing, including parametric notes, view scale notes, tables, symbols, and cosmetic thread notes.

When dual dimensioning is used, controls the use of leading and trailing zeros in both standards independently.

If the units in the "dual_dimensioning" drawing setup file option are "primary[secondary]," "std_english[std_metric]" shows the primary units values with trailing zeros, while the secondary units show values with leading zeros.

If the units in the "dual_dimensioning" drawing setup file option are "secondary[primary]," "std_english [std_metric]" secondary units show values with trailing zeros, while the primary units show values with leading zeros.

If set to "std_default," displays the dimension or parameter according to its units. If set to "both," displays both leading and trailing zeros in dimensions or parameters, regardless of whether metric or English units are used.

lead_trail_zeros_scope

dims, all

Controls whether only dimensions are affected by the setting of the drawing setup option "lead_trail_zeros." If set to "dims," the drawing setup option "lead_trail_zeros" controls only dimensions. If set to "all," the drawing setup option "lead_trail_zeros" controls dimensions and also all parameters, including parametric notes, view scale notes, tables, symbols, and cosmetic thread notes.

line_style_length

font_name default, font_name value

Sets the length of elements composing a font. You must add this option to the drawing setup file whenever you want to modify the length. Type the font name and then a desired value for the font length in system units. The "default" setting indicates default length values.

line_style_standard

std_ansi, std_iso, std_jis, std_din

Controls text color in drawings. Unless set to "std_ansi," displays all drawing text in blue, and displays boundary of detailed views in yellow.

location_radius

default (2.), 0.0, any value

Modifies radius of nodes indicating location, improving their visibility, particularly when printing drawings. If set to "default," sets radius as 2 drawing units. If set to "0.0," displays location nodes, but does not print them.

There is no maximum value for this setting.

max_balloon_radius

0.000000, non-zero , value

Sets the maximum allowable balloon radius. If set to "0," balloon radius depends only on text size.

mesh_surface_lines

on, off

Controls display of blue surface mesh lines.

min_balloon_radius

0.000000, non-zero value

Sets minimum allowable balloon radius. If set to "0," balloon radius depends only on text size.

model_digits_in_region

yes, no

Controls display of number of digits in two-dimensional repeat regions. If set to "yes," two-dimensional repeat regions reflect the number of digits of part and assembly model dimensions.

model_display_for_new_views

default

Specifies the view display for hidden lines.

model_grid_balloon_size

0.200000, value

Specifies default radius of balloons shown with the model grid in a drawing.

model_grid_neg_prefix configuration file option

- , any string

Controls prefix of negative values shown in model grid balloons.

model_grid_num_dig_display

0, value (integer)

Controls number of digits displayed in grid coordinates that appear in grid balloons. Type an integer specifying the number of decimal places, or use the system default (0) to display coordinates as integers.

model_grid_offset

default, value

Controls offset of new model grid balloons from the drawing view. If set to "default," offsets new model grid balloons from the drawing view by twice the current model grid spacing. If set to a value, offsets balloons by that number of inches (not drawing units) from the view.

new_iso_set_datums

yes, no

Controls display of set datums. If set to "yes," displays set draft datums in accordance with the ISO standard.

node_radius

default, value

Controls display of nodes in symbols. If set to "default," system specifies radius of nodes. If the setting is so small that the nodes do not appear, the system uses the default setting. There is no maximum value for this setting.

ord_dim_standard

std_ansi, std_iso, std_jis, std_din

Sets the standard according to the display of ordinate dimensions. If set to "std_ansi," shows dimensions without a connecting line (Figure a). Otherwise, places related ordinate dimensions along the connecting line that is perpendicular to the baseline and starts with a circle (Figure b). Each segment of the connecting line ends with an arrow. Displays arrows and the circle filled or open, according to the current setting of the drawing setup file option "draw_arrow_style." Note that when witness lines are interconnected, moving any of the related dimensions moves all of them.
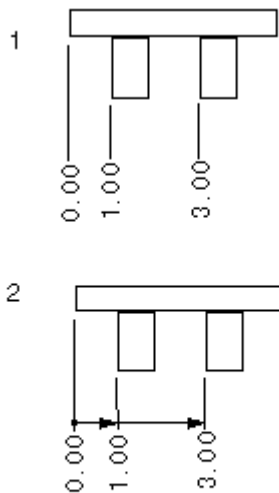
1) If set to "std_ansi"

2) If set to "std_iso," "std_din," or "std_jis."

orddim_text_orientation

parallel, horizontal

Controls ordinate dimension text orientation. If set to "parallel," displays dimension text parallel to the leader lines. If set to "horizontal," displays it horizontally, parallel to the bottom of the drawing sheet.

parallel_dim_placement

above, below

Determines whether dimension value displays above or below leader line when you set the "text_orientation" option to "parallel." This option does not apply to dual dimensions.

pipe_pt_shape

cross, dot, circle, triangle, square

Controls shape of theoretical bend intersection points in a piping drawing.

pipe_pt_size

default, value

Controls size of theoretical bend intersection points in a piping drawing.

projection_type

third_angle, first_angle

Determines method for creating projection views.

radial_pattern_axis_circle

no, yes

Sets display mode for axes of rotation that are perpendicular to the screen in radial pattern features. If set to "no," displays axis lines (Figure a*)*. If set to "yes," a circular shared axis appears, and axis lines pass through the center of a rotational pattern (Figure b).

ref_des_display

no, yes. DEFAULT

Controls display of reference designators in a drawing of a cabling assembly. If set to "DEFAULT," selects the **Reference Designators** checkbox in the Environment dialog box.

remove_cosms_from_xsecs

total, all, none

Controls display of datum curves, threads, cosmetic feature entities, and cosmetic cross-hatching in a full cross-sectional view. If set to "all," removes datums and cosmetics from all types of cross-sectional views. If set to "total," removes features located entirely in front of the cutting plane from the cross-sectional view. Displays these features in full only if they intersect the cutting plane. If set to "none," displays all datum quilts and cosmetic features.

restricted_gtol_dialog

Yes (default value) No

This config controls the restrictions in the Geometric Tolerance Dialog. When set to YES (the default), the dialog will adhere to standards when picking certain GTOL types. This is current behavior. Setting the value to NO will cause the dialog to drop all restrictions. select_hidden_edges_in_dwg
yes, no

Disallows the selection of No Hidden edges in drawings when using Query Select by rejecting edges behind the first surface at the selection point.

show_cbl_term_in_region

no, yes

Allows use of the report symbols "&asm.mbr.name" and "&asm.mbr.type" to show terminators in Pro/REPORT tables for cable assemblies having connectors with terminator parameters. If set to "yes" (you must set the attribute **Cable Info** for the repeat region), shows terminators. When creating new drawings, the default value is "yes." For existing drawings, the default value is "no."

show_pipe_theor_cl_pts

bend_cl, theor_cl, both

Controls display of centerlines and theoretical intersection points in piping drawings. If set to "bend_cl," shows centerlines with bends only. If set to "theor_cl," shows only centerlines with theoretical bend intersection points. If set to "both," shows both bends and theoretical intersection points.

show_preview_default

remove, keep
Determines the default behavior for preview in the Show/Erase dialog. Set to Remove sets the default to be Sel to Remove. Set to Keep sets the default to be Sel to Keep.

show_quilts_in_total_xsecs

no, yes
Determines if surface geometry is included in a drawing cross section and whether the surface will be cut by the cross section cutting plane. Set to no (default) to exclude surface geometry. Set to yes to include surface geometry.

show_total_unfold_seam

yes, no

Controls display of seams (the edges of the cutting plane) in total unfolded cross-sectional views. If set to *yes*, makes the seams; if set to "no," blanks them.

shrinkage_value_display

percent_shrink, final_value

Displays dimension shrinkage in percentages or as final values.

sym_flip_rotated_text

no, yes

Flips any text in a **Rotate Text** symbol that is upside down, making it right-side up. If set to "yes" and the symbol orientation is +/- 90 degrees, flips the text, rotating it along with the symbol (for an illustration, see Manipulating Instances on page 10 - 28).

sym_rotate_note_center

yes, no

Controls display of a symbol and its notes and their reorientation upon mirroring and rotation. If set to "yes," rotates symbol notes as if their origin was not at the bottom of the text, but in the middle of the height of the text instead. If set to "no," rotates text by rotating its origin point, as it is.

tan_edge_display_for_new_views

default
Specifies the tangent edge display.

text_orientation

horizontal, parallel, parallel_diam_horiz, ISO_parallel, ISO_parallel_diam_horiz

Controls orientation of dimension text in the drawing. If set to "horizontal," displays all dimension text horizontally. If set to "parallel," displays text parallel to a dimension leader line. If set to "parallel_diam_horiz," displays all dimensions except diameter dimensions parallel to their leaders; displays only *diameter* dimensions horizontally. The elbow of a diameter dimension always extends through to the end of the text. To control the elbow extension beyond the text, use the drawing setup file option "dim_text_gap." If set to "ISO_parallel," displays text parallel to a dimension witness line. This differs from the value "parallel" since this will also allow dimension tolerances to be displayed in ISO 406:1987 (E) format or British Standard Tolerance Format. If set to "ISO_parallel_diam_horiz," displays all dimensions except diameter dimensions parallel to their leaders. Limit tolerances will be stacked in accordance with ISO standard.

The drawing setup file option "angdim_text_orientation" (not "text_orientation") controls the display of angular dimensions.

text_thickness

0.000000, 0<value<.5

Sets default text thickness for new text after regeneration and existing text whose thickness has not been modified. Type the value in drawing units.

text_width_factor

0.800000, .25<#>8

Sets default ratio between the text width and text height. The system maintains this ratio until you change the width using the **Text Width** command.

thread_standard

std_ansi, std_ansi_imp, std_iso_imp, std_iso_imp_assy, std_ansi_imp_assy, std_iso

Controls display of threaded hole with an axis (perpendicular to the screen as an arc (ISO standard) or as a circle (ANSI) standard).

If set to "std_ansi_imp" or "std_iso_imp," does not display hidden thread lines when you select **No Hidden Line** from the **Display Style** list in the Pro/ENGINEER Environment dialog box. When you select **Hidden Line**, displays thread lines as leader lines (yellow). If set to "std_iso_imp_assy," displays threads in cross-sectional assembly drawings in accordance with the ISO 6410 standard. If set to std_ansi_imp_assy," displays them in accordance with the ANSI standard. The "std_iso" and "std_ansi" values are valid for drawings created before Release 15.0.

tol_display configuration file option

no, yes

Controls display of dimension tolerances. You cannot access the Pro/ENGINEER Environment dialog box when this option is set.

tol_text_height_factor

standard, number > 0

Sets default ratio between the tolerance text height and dimension text height, when showing the tolerance in "plus-minus" format. If set to "standard," the system uses 1 for the ANSI standard and .6 for the ISO standard.

tol_text_width_factor

standard, number > 0

Sets default factor to maintain a proportion between the tolerance text width and dimension text width, when showing the tolerance in "plus-minus" format. If set to "standard," the system uses .8 for the ANSI standard and .6 for the ISO standard.

use_major_units configuration file option

default, yes, no

Controls whether fractional dimensions are measured in feet and inches. Controls the display of fractional dimensions in drawings in the same way that the configuration file option `use_major_units` controls the display of fractional dimensions in Part and Assembly modes.

If set to "no," fractional dimensions are not displayed in major units.

If set to "yes," fractional dimensions are displayed in major units.

If set to "default" (the default setting), displays fractional dimensions according to the setting of the configuration file option `use_major_units`.

view_note

std_ansi, std_din, std_iso, std_jis

If set to "std_din," creates a view-related note with the words "SECTION," "DETAIL," and "SEE DETAIL" omitted.

view_scale_denominator

0, integer

Determines denominator for the view scale before simplifying the fraction. If set to a positive integer and `view_scale_format` is a decimal, the view scale chosen for the first view of a model in the drawing is rounded to the nearest value with the specified denominator. If the view scale is so small that rounding would make the scale 0.0, the value of `view_scale_denominator` is automatically changed by multiplying it by the smallest power of 10, which would give a positive value if rounding the scale down (although rounding up can happen). When you type a view scale value, you can round it to an allowable fraction. The system does not round existing scale values after you have edited the setup file; they are approximate. Displays approximate scales preceded by a tilde (~), if you set the configuration file option `mark_approx_dims` to `yes`. If you set it to `0`, expresses the scale value in decimal format.

view_scale_format

decimal, fractional, ratio_colon

Expresses a scale as a decimal or fractional value. If set to "ratio_colon," displays the view scale values as a ratio. For example, instead of a view scale of 0.5, displays the view scale as 1:2. Since the ratio is just another way of displaying a fraction, make sure that you set the "view_scale_denominator" option appropriately.

weld_solid_xsec

weld_symbol_standard

std_ansi, std_iso

Displays weld symbols in a drawing according to the ANSI or ISO standard.

witness_line_delta

0.125000, value

Sets the extension of the witness line beyond the dimension leader arrows.

witness_line_offset

0.062500, value

Sets offset between a dimension line and object being dimensioned. This gap may be visible only when you plot a drawing. To see the effect, use the screen plot.

Also controls the size of the break at the intersection of witness lines, when you use the "Dimension" type break.

yes_no_parameter_display

true_false, yes_no

Controls display of "yes/no" parameters in drawing notes and tables. When set to "yes_no," parameters can have a "yes" or "no" value in drawing notes. When set to "true_false," they can have a "true" or "false" value.

About Layers in Drawing Mode

In Pro/ENGINEER, a *layer* is a grouping of draft items (draft dimensions, sketches) as well as model items such as features and datum planes. You may want to group items for several reasons (for example, for manufacturing purposes only or for separate verification). The main reason for using layers is to blank construction geometry (such as surfaces and datum curves) that you do not need to show in your drawing.

Although layers can represent different entities, they always have the following three properties in common: *name*, *display status*, and *contents*. The display status (shown, blanked, or isolated) controls whether or not items in the layer are displayed. Layers can include other items in a Pro/ENGINEER database such as features, dimensions, notes, geometric tolerances (gtols), and other layers. You can place items on layers using two basic methods: by copying them from other drawing, part, or assembly layers, or by selecting them directly from the drawing.

Manipulating Layer Display Status in Individual Views

Drawing views do not have individual layers for each view—they use the layers in the drawing. However, Pro/ENGINEER can control the layer display status of drawing layers separately for each view. You can display drawing layers independently for individual drawing views or make the layer display dependent on the drawing.

By setting the display of drawing layers independently of the models and individually for each view, you can do the following:

- Include items from parts and assemblies directly in a drawing layer without including them on a layer in a model.
- Control the display of drawing layers to include items from the model without marking the model as changed.
- Vary the display of a single model in different drawing views.

If you specify independent view display, it takes precedence over the main drawing for that view. When you create an independent view, the layer display defaults to that of the main drawing. When you create a detailed view, the layer display defaults to that of the parent view. You can then modify the display for any view independently, or reset it to follow the drawing display. You can copy the display status for the drawing or a view to any other view or the drawing, and then modify the display individually for each.

To Add Items To A Drawing Layer

Using the **Add** command on the Item menu in the Layers dialog box, you can add an item to a layer.

1. Choose **Layers** from the Pro/ENGINEER View menu. The **Layers** dialog box opens.
2. Select one or more layers in the Layer Tree.
3. Choose **Item** > **Add** (or click the toolbar shortcut or use the RMB). The CONFIRMATION prompt appears, asking whether you want to add the specified layer to the active drawing.
4. Click **Yes**, and the LAYER OBJ menu appears with a list of possible item types.
5. Choose one or more item types and specify item selections, using the menus that appear for each type of item.
6. Select the particular items to place on selected layers:
   - If the drawing is Independent, you can select model items to place on a drawing layer. Select items in the graphics window, or in the Model Tree, or by navigating through the menu structure.
   - If the drawing is Dependent, you can place only drawing items on a drawing layer. Select items in the graphics window or by navigating through the menu structure.

To Change the Display of Layers in a Drawing View

1. Choose **Layers** from the Pro/ENGINEER View menu. The **Layers** dialog box opens.
2. In the **Layers** dialog box, select **Drawing View** from the **Active Object** list. The system highlights the view outlines.
- The layer display of the view outlines shown in cyan is *dependent* on the layer display in the drawing.
- The layer display of the view outlines shown in green is *independent* of the layer display in the drawing.
3. Click on a view to select it and manipulate its layer display. As soon as you select the view, its layer display becomes independent of the drawing, and the system updates the **Active Object** window with the name of the selected view.
4. Use the dialog box to change the display of various layers in the drawing view. When you have finished working with the view, select another object from the **Active Object** list in which to manipulate layer display.

To Ignore the Layer Status in a Drawing Model

Using the **Layer Status Control** dialog box, you can control the display of model layers in a drawing, without having to make any change to the part or assembly in which that item was created.

You can force a drawing to ignore the layer status in its model completely when determining if it should display an item on a layer. The system displays all items on model layers in the drawing, and you can manipulate them separately at the drawing level. You can blank or display layers at the drawing level without making changes to the part or assembly in which the item was created, and the model does not change. To set a drawing so that it ignores the layer status of its model, do one of the following:

- Choose **Preferences** from the **Status** menu in the **Layers** dialog box, and select the first check box (**Ignore display status of layers in the model**) in the **Layer Status Control** dialog box.
- Set the drawing setup file option `ignore_model_layer_status` to `yes` (the default value). If you set

this option to `no`, the drawing layer status follows that of the model.

Tip: Modifying Drawing Layers does not Affect the Model

Modifying the display of a layer in a drawing does not mark any parts or assemblies as modified. Therefore, you do not need to save parts and assemblies and resubmit them to Pro/PDM when you toggle layers and save them. Also, the drawing looks as it should when you retrieve it, regardless of the changes that you may have made to the model layers. Because one of the benefits of the `ignore_model_layer_status` setting is that changes will not be made to the model or model layers, you cannot add items or remove them from model layers. However, you can manipulate items directly on drawing layers.

Invisible Drawing Model Items

If an item in the part or assembly does not have its display set in any way from the drawing, it shows by default. Items in the drawing model are visible by default unless one of these conditions is met:

- The item is included directly on a drawing layer.
- The item is on a part or assembly layer with the same name as a drawing layer and the drawing setup file option `draw_layer_overrides_model` is set to `yes`.
- The item is on a part or assembly layer which has been included in a drawing layer.

For these three cases, the system determines whether to show the item based on the drawing layer status information.

To Control Model Layers with the Same Name from the Drawing

You can set the display of drawing model layers to follow the display of drawing layers with the same name. For example, if you blank the drawing layer `datums`, the system also blanks all items on the `datums` layers of all drawing model components. If the model is an assembly and you then add another component with a layer of the same name, it automatically sets the display of any items on a model layer named `datums` to be the same as the drawing layer. To keep one component `datums` layer from being affected, put the desired layer directly on another drawing layer. To set the display, do one of the following:

- Choose **Preferences** from the **Status** menu in the Layers dialog box, and select the second check box (Change display of model layers with the same names in drawing only) in the **Layer Status Control** dialog box.
- Set the drawing setup file option `draw_layer_overrides_model` to `yes`. If you set this option to `no` (the default value), the system ignores nondrawing layers when you set the display status of layers in the drawing model with the same name.

Using the Layer Status Control Dialog Box

When using the Layer Status Control dialog box to manipulate layer display status, keep in mind the following:

- If you do not select the first check box (**Ignore display status of layers in the model**), the second check box (**Change display of model layers with the same names in drawing only**) is unavailable. You can then see changes to model layers in the model in the drawing.
- When you do not select any check boxes, if you blank any of the part layers in the drawing, the system blanks the items in the drawing and in the part when you retrieve the part.

To Make the Display of a Drawing View Dependent on the Layer Display

1. Choose **Layers** from the Pro/ENGINEER **View** menu. The **Layers** dialog box opens.
2. In the **Layers** dialog box, select **Drawing View** from the **Active Object** list; then select a drawing view with independent layer display. When the active object is an independent drawing view, the Make Dependent toolbar shortcut is available.
3. Change the drawing view to dependent using either of the following methods:
   - Click the Make Dependent toolbar shortcut.
   - Choose Status > **Preferences**, and in the **Layer Status Control** dialog box uncheck the first option, **Ignore display status of layers in the model**.
4. When the system asks you if you really want the layer status of the drawing view to be dependent on the layer status of the drawing, click **Yes**. The system updates the drawing and the **Active Object** returns to **The Drawing**.

To Switch Items from One Layer to Another

Using the **Switch** command on the Item menu in the **Layers** dialog box, you can move items from one layer to another in the active drawing. The **Switch** command is available when nothing is selected in the Layer Tree or when at least one layer item is selected in the Layer Tree.

1. Choose **Layers** from the Pro/ENGINEER View menu. The **Layers** dialog box opens.
2. Select one or more layer items in the Layer Tree.
3. Choose **Item > Switch** (or use the RMB). The **Select Layers** dialog box opens, listing available layers in the active drawing. The source layer (the layer that contains the item to be switched) is highlighted.
4. Specify destination layers for the item(s). Only layers from the item's parent object are available. You can leave the item in its original layer and add it to several other layers.
   - To remove the item from a layer in which it is already present, clear the layer name.
   - To add the item to a layer in which it is not already present, select the layer name.

   If the item is not blanked, it is highlighted in green.

   If the item is a blanked geometry feature (such as a hole or round), or a blanked component, the geometry is still visible on the screen, but remains highlighted in *red*. The system tells you through the message area that the item is blanked.

   If the item is blanked and is not visible on the model (such as a datum or axis), the item is not highlighted on the model at all.

Notes:

- You can select items that are shown or blanked. However, you cannot select suppressed items.
- If the items are visible on the model (whether or not they are blanked), they are highlighted in red after you select them.
- If you select a blanked component in an assembly, the component appears on the assembly highlighted in red.
- If items such as datums and axes are blanked, they are not visible on the model and so are not highlighted.

   If the items are not blanked, the system highlights the first item you selected, in green, on the model and displays its feature number and type in the message area, for example, feature 6 (ROUND). The LAYER SEL menu appears with a list of the layers in the active model. The layers in which the item is already present (if any) are highlighted.
5. Select **Done Sel**. The system clears the item (if it was previously present) from all the layers that are not selected and adds it (if not already present) to all the layers that are selected.
6. Repeat for each additional item, if you selected more than one.
7. When you have finished, choose **Done/Return**.

Tip: Adding Items

If you have many items, you can group them in a layer and then copy that layer as an item to other layers.

To Copy Items from One Layer to Another

Using the **Copy** command on the Item menu in the **Layers** dialog box, you can copy items from one layer to another.

1. Choose **Layers** from the Pro/ENGINEER **View** menu. The **Layers** dialog box opens.
2. Select one or more layer items in the Layer Tree.
   You can copy an entire layer by selecting it as an item to be copied.
3. Choose **Item** > **Copy** (or use the RMB). The item(s) are copied to the clipboard.
4. Select one or more of the existing layers in the Layer Tree, or choose **New Layer** to create a new layer.
5. With all destination layers highlighted, choose **Item** > **Paste** (or use the RMB). The item(s) are placed on the layers.
   Note: Copying the items from one layer to another is a one-time task. If you add another item to the original layer, the system does not add the item to the destination layer.

To Save a Model Layer to Disk

1. Choose **Layers** from the Pro/ENGINEER **View** menu. The **Layers** dialog box opens.
2. In the model, choose File > **Save Status File**. The system saves to a file the layers of the active model and their current status.
   You can use these layers and their display status for other objects, or retrieve them later.
   Notes:
   If you enter the name `layer_file` in response to the prompt, the actual filename is `layer_file.pro`.
   The system does not save display changes with the model when it saves the model unless you click **Save Status** before you exit the **Layers** dialog box.

To Replace Models

The **Select Instance** dialog box is available when you are replacing the model of one family table instance with another. This dialog box displays all the available instances and allows you to replace the current drawing model with another model in the same family, without having to remember and type the name of the replacement instance.

If a model that you use in a drawing is a member of a family of parts or assemblies, you can replace that model with other family members. The system maintains dimensions, attached notes, and other drawing annotations when one model replaces another.

Use the **Select Instance** dialog box to select an instance with which to replace the model.

1. Choose **VIEWS > Dwg Models > Replace**.
2. Select the model that you want to replace as the active model. The **Select Instance** dialog box opens when you are replacing the model of one family table instance with another.
3. Select the name of another family member to replace it.
4. The **Select Instance** dialog box displays a list of all instances of the currently selected family table model. You can select one of these to use as the replacement model. You can select an instance by name when the **By Name** page is visible (the default), or click the **By Parameter** tab to select an instance by parameter.
5. After selecting an instance to be the replacement, click **Open**. The current model is replaced by the selected instance.
   Note: When you are using automatic replacement, if the system shows dimensions in the assembly drawing on a component that you have replaced with the new instance, it displays equivalent dimensions. However, if you replace the component using manual replacement, it does not preserve the dimensions.

If a model that you use in a drawing is a member of a family of parts or assemblies, you can replace that model with other family members. The system maintains dimensions, attached notes, and other drawing annotations when one model replaces another.

To Enlarge an Area of a Drawing

1. Choose View > **Pan/Zoom...**.
2. Define the area to enlarge by using the mouse to enclose it in a box. The system displays the enclosed area.
Zooming out on the drawing doubles the area of the drawing currently shown on the screen. To reduce an area of a drawing, use **Zoom Out** in the PAN-ZOOM menu. Use the **Reset** command to resize the model so that the entire model fits within the screen, keeping the current view orientation. If you perform a pan on a drawing, you can select a new center for the screen.

To Pan the Center of the Screen

1. Choose PAN-ZOOM > **Pan**.
2. Select a location on the screen to become the new screen center.
   Pro/ENGINEER shifts the display of the drawing on the screen so that the selected point is the center of the screen display.

In Drawing mode, you can save, retrieve, and delete pan-zoom settings on drawing views. To save a pan-zoom view, choose **Saved Views...** from the **View** menu, followed by **Save**. To retrieve a pan-zoom view, select its name from the Saved Views dialog box; to delete a view, click **Delete**.

To Show Multiple Windows

By choosing **New** from the Pro/ENGINEER **Windows** menu, you can show multiple drawing sheets in multiple windows (or the same sheet in multiple windows), and you can select any of them.

To Change the Display of Selected Views

To change the display of selected views, assembly components, or even individual edges, use **Disp Mode** in the VIEWS menu. Note that the changes you make to the display mode using this command override the global settings in the **Environment** dialog box.

Note: When you have set the display mode of views and edges to **Hidden Line,** they appear in phantom font on the screen but are plotted as dashed lines.

You can assign a color to datum curve segments by choosing **Model Display...** from the Pro/ENGINEER **View** menu and **With Datum Curves** from the **Shade** page of the Model Display dialog box.

View Display

By default, the drawing view display appears as specified in the **Environment** dialog box. During a Pro/ENGINEER session, you can modify the default display. However, keep in mind that this changes the display of all of the views in the drawing.

About Detail Items

If you have a license for Pro/DETAIL, you can create draft dimensions on a view. In Drawing mode, you can also show dimensions of a model in a drawing.

You can include notes, symbols, and geometric tolerances in a drawing.

Manipulating Detail Items

Pro/DETAIL commands let you manipulate dimensions, notes, geometric tolerances, surface finish symbols, cross-section arrows, set datums, and dimension text in the following ways:

- Move them (within a view or between views).
- Reattach leaders to another entity.
- Add more leaders.

- Move jogs.
- Translate them.
- Remove them (erase) from the display.
- Display them.
- Delete them.
- Modify the values.
- Relate them to a view.
- Relate them to dimension text

To Create a Draft Dimension

1. Click **Insert > Dimension**, and then select **New Refs…, Common Refs…**, or **Ordinate…**.
2. Select two entities or edges on the view.
3. Place the dimension using the middle mouse button.

To Move Dimensions of Draft Features

1. Click to select the item you want to move. The cursor changes to a four-pointed arrow. The dimension is attached to the arrow.
2. Drag the mouse to move the dimension. Click again to place the dimension.

To Show or Hide Drawing Dimensions

1. Click **View** > **Show and Erase**.
2. In the **Show/Erase** dialog box, click **Dimension** in the **Type** box.
3. To specify the dimensions as ordinate, click **Options** and select **Switch to Ordinate**.
4. Click **Pick Bases** and select a baseline.
5. Select **Feature**, **Feat & View**, or **View**. To show all dimensions for the current model, click **Show All**.
6. To preview what the drawing will look like when you make the changes, click **Preview** and specify items to show.

To Move Detail Items

1. Click to select the object you want to move. To select multiple items, hold down the Shift key as you select. The object is highlighted and the cursor changes to a four-pointed arrow.
2. Drag the object to the new location.
3. Click again to position the object.

To Rotate Detail Items

1. Click **Tools > Rotate.**
2. Select the detail item or items to rotate. Click **Done Sel**.
3. Use the options on the **Get Point** submenu to pick a rotation point. You sre prompted to enter a degree of counterclockwise rotation around the point.
4. Enter the degree number. Press **Enter**. The item is rotated

To Move an Item Dynamically

In drawings, when you move an item dynamically, the item dims to gray (marking the item's original location). As you move it, the system highlights it in red (all text appears as boxes to enable faster speed in moving the items). Use the left mouse button to set the item in a new location; use the middle mouse button or a menu command to return the item to the original location; and use the right mouse button to flip dimension arrows. All statements in parentheses following each bulleted item below explain how you can perform these actions using the menu selections.

You can perform the following tasks using dynamic move:

- Move jogs in both witness and leader lines.
- Move dimensions, notes, geometric tolerances, draft entities, draft groups, symbols/surface finishes, axes, datum tags, and Bill of Materials (BOM) balloons.
- Move the text of dimensions, notes, geometric tolerances, datum tags, symbols/surface finishes, and axes.
- Move the attachment of radial, chamfer, and constant draft feature dimensions; notes; geometric tolerances; symbols/surface finishes; and BOM balloons.
- Align linear and ordinate dimensions.
- Skew the display of linear dimension witness lines and select the ends of the witness lines closest to the text.
- Clip dimension witness lines, datums, and axes The dimension does not appear in the original position during the move.
- Change the location of a table within a single sheet of a drawing. The system erases the original table and highlights the table in white.

To Move an Item Between Views

You can move items attached to the model with leader lines (or attached directly to an edge) from one view to another of the same model.

1. Select an item to move.
2. Click **Edit** > **Switch to View** (or click **Switch View** in the right mouse button pop up window.)
3. Select the view to which you want to move the item. The item is attached to the new view, and activated to move for adjustment.

To Attach a Leader to a New Object

1. Click to select a leader to modify.
2. On the right mouse button pop up menu, click **Mod Attach**.
3. Select a new entity at the point to attach the leader. Use the middle mouse button to set the leader in its new placement.

To Add a New Leader

1. Select the item to which you would like to add the leader.
2. From the right mouse button pop up menu, click **Mod Attach.**
3. In the Menu Manager click **MOD OPTIONS** > **Add Ref**.
4. Select a reference point on the view for the leader. You can select several points if you want to.
5. Click the middle mouse button, or choose GET SELECT > **Done Sel** and ATTACH TYPE > **Done**. The leader or leaders are attached at the specified points.

To Insert or Move Leader Jogs

To insert the jog

1. Click to select the leader.
2. From the right mouse button pop up menu, click **Make Jog**.
3. Click to select the jog point. The jog is inserted, and is attached to the cursor for placement.
4. Click the left mouse button to complete. The leader is still selected, you can select another point to put a jog.
5. Place another jog, or left click again to exit the command.

To Move Unattached Entities

To move draft entities and annotations not attached to an edge, use the **Translate** command. Unlike the **Move** command, you can translate any number of items at one time.

1. On the Menu Manager, click **DRAWING** > **Tools > Translate**.
2. Select the draft items to translate; then choose **Done Sel**.
3. Define the translation vector by selecting the endpoints with the left mouse button. The endpoints

correspond to the initial and final locations of a reference point on the screen. The system translates selected items the distance equal to the length of the vector in the same direction as the vector.

About Displaying Cosmetic Features

Pro/ENGINEER displays cosmetic features (except cosmetic threads) in views unless their sketching plane is perpendicular to the screen. You can control the display of threads in a drawing using the drawing setup file option `hlr_for_threads`. If you set it to `yes`, thread edges meet the ANSI or ISO standard for hidden line display (set by the `thread_standard` option). If you set it to `no`, the system displays thread edges as surfaces as they would be in Part mode.

You can show or erase sketched cosmetic features, weld features, and cosmetic threads in a drawing view without having to show or erase datum planes as well.

Note: The erasure or display of these items in a detailed view is always the same as that of the parent; you cannot modify it individually.

To Show or Hide Cosmetics

To show cosmetic features, choose **View** > **Show and Erase**, then click **Show** and **Cosmetic Feature** in the Show/Erase dialog box. To erase them, click **Erase** and **Cosmetic Feature**. You can entirely erase cosmetic threads from a drawing view.

Note: Click **Note** in the **Type** box of the Show/Erase dialog box to erase pattern notes, the notes that display the number of members of a pattern (2 HOLES, for example).

To Show Threads in Cross-Sectional Assembly Views

To display threads in cross-sectional assembly drawings according to the ISO or ANSI standard, use the drawing setup file option `thread_standard` in conjunction with `hlr_for_threads`.

- To display them in accordance with the ANSI standard, set the drawing setup file option `thread_standard` to `std_ansi_imp_assy`.
- To display them in accordance with the ISO 6410 standard, set it to `std_iso_imp_assy`.

Note: In the ISO-standard display, if the bolt diameter equals the hole thread diameter, and the bolt thread diameter equals the hole diameter, the bolt covers the hole's crosshatch if one of these conditions exists:

- The bolt and hole are coaxial.
- Both axes are in the cross section.
- The cross section itself is planar and parallel to the screen (normal to the viewing direction).

In the case of an offset cross section, the portion of the cross section that intersects the bolt/hole pair must be planar and parallel to the screen.

To Delete a Draft Item

Use the **Delete** command, to remove reference dimensions, notes, and geometric tolerances from a drawing, as well as unset any reference datums.

1. Select the item to delete from the drawing. Hold down the shift key to select multiple items.
2. Click **Edit** > **Delete**.
   Note: Using the **Delete** command on a set datum or on a basic dimension removes gtol referencing for that item (but does not delete it).

To Modify a Note or Dimension Value

You can modify the numeric values within geometric tolerances and surface finishes, as well as dimensions. When modifying the value of dimensions, however, you must regenerate the drawing before the system updates the part or associative draft dimensions to the new value.

1. Click **Edit > Value**.
2. Select the value to modify.
3. Type a new value. Press Enter.

To Associate Detail Items with a Drawing View

You can associate detail items (such as draft entities, notes, and symbols without leaders) with a view.

1. Choose **VIEWS > Relate View > Add Items**.
2. Select the view to which you want to associate detail items.
3. Select the detail items to associate; then choose **Done Sel**.

Note: When using this command, keep in mind the following:

- You cannot relate draft datums to a model view.
- If you create a note or symbol using the **Free Point** command, you cannot relate it to a view. The system attaches a standard note (one with a leader) to the view or draft entity.
- Draft items related to a drawing view take on the same behavior as the view. When you perform actions on the view such as moving, erasing, or deleting, the system performs the same actions on the related draft items.

To Dissociate Detail Items from a Drawing View

1. Choose VIEWS **> Relate Views > Remove Items**.
2. Select the items to dissociate; then choose **Done Sel**.

After associating items with a view, you can use the Pro/ENGINEER **Info** menu to identify the entities in any view or in a specific view.

To Highlight Draft Entities

1. On the Pro/ENGINEER menu bar, click **Info** > **Drawing > Highlight by Attributes**.
2. In the **Highlight By Attributes** dialog box, select **2D Sketched Entity** from the **Item Type** box.
3. Do one of the following:
   - Select **Any** from the **Displayed in View** box to highlight attributes in any view.
   - Select **Selected** and click **Select View** to choose a specific view.

   The system highlights all of the currently displayed items that have been associated to the view or views specified.

To Relate Draft Objects to Dimensions

After you create a note, draft geometric tolerance, or symbol, you can relate it directly to dimension text so that it moves with the dimension when the dimension changes location. You can relate these detail items to a dimension when specifying its placement.

1. Choose **Tools > Relate Obj > Add Items**.
2. Select a dimension to which the system should relate the items.
3. Select notes, symbols, or draft gtols to relate to the selected dimension.
4. To remove them, choose **Remove Items** from the Relate Obj menu.

Note: When you erase, redisplay, or delete a dimension, the system also erases, redisplays, or deletes all items related to it.

About Note Parameters

In addition to text and special symbols, notes can include model dimensions, pattern instance parameters, symbols, drawing labels, and drawing parameters. The drawing setup file option `yes_no_parameter_display` controls the display of `yes`/`no` parameters in drawing notes and tables.

To Include Parameter Information in Notes

When you set the drawing set up file option `yes_no_parameter_display` to `yes_no`, parameters can have a `yes` or `no` value in drawing notes. When you set it to `true_false` (the default value), they can have a `true` or `false` value.

To specify parameter information in a note, use the following format:

- Dimensions—`&d#` or `&ad#`, where # is the dimension ID. Examples: `&d12`. `&ad5`.
- Reference dimensions—`&rd#`, where # is the dimension ID. Example: `&rd2`.
- Instance number—`&p#`, where # is the parameter ID. Example: `&p8`
- User-defined parameters—`&xxxx` where `xxxx` is a symbol defined in a relation.
- Datum names—`&dtm_name`, where name is the name of a datum plane.
- Drawing labels—You can add the following drawing labels to a drawing:
  - `&todays_date`—Adds the date as of the note's creation in the form dd-mm-yy (for example, 2-Jan-92). You can edit it later as any other nonparametric note, using **Text Line** or **Full Note**. If you include this symbol in a format table, the system evaluates it when it copies the format into the drawing.
  - `&model_name`—Adds the model used for the drawing.
  - `&dwg_name`—Adds the name of the drawing.
  - `&scale`—Adds the scale of the drawing.
  - `&type`—Adds the drawing model type.
  - `&format`—Adds the format size.
  - `&linear_tol_0_0` through `&linear_tol_0_000000`—Adds the linear tolerance values for one to six decimal places.
  - `&angular_tol_0_0` through `&angular_tol_0_000000`—Adds the angular tolerance values for one to six decimal places.
  - `&current_sheet`—Adds the sheet number for the sheet on which the note is located.
  - `&total_sheets`—Adds the total number of sheets for the drawing.
- Drawing parameters—`&parameter:d`, where parameter is the parameter name.

Considerations when Specifying Parameter Information in a Note

When specifying parameter information in a note, keep in mind the following restrictions:

- When you include dimensions in notes, the system removes them from their original views.
- You cannot add draft dimensions (associative—add # or nonassociative -dd#) to a note.
- A datum name in the note is read-only, so you cannot modify it; unlike dimensions, a datum name does not disappear from the model view if it is included in a note. The system encloses its name in a rectangle, as if it were a set datum.
- You cannot use drawing labels in drawing relations; you can only use them in drawing notes and tables.
- When the note appears, the system replaces dimensions and parameters by the corresponding numerical values. If you set the configuration file option `switch_dims_for_notes` to `yes`, dimensions appear as their symbolic values during note creation. If you set it to `no`, they remain as numerical values.
- Before referencing a model in a parametric note, make sure that you add this model to the drawing.

Otherwise, the system does not evaluate the model parameter referenced in the note.

• You can include a gtol symbol in a note by entering it as a parameter; however, you cannot include global parameters (that is, parameters created in a layout) and draft dimensions in notes.

Example: Parameter Symbols for Notes
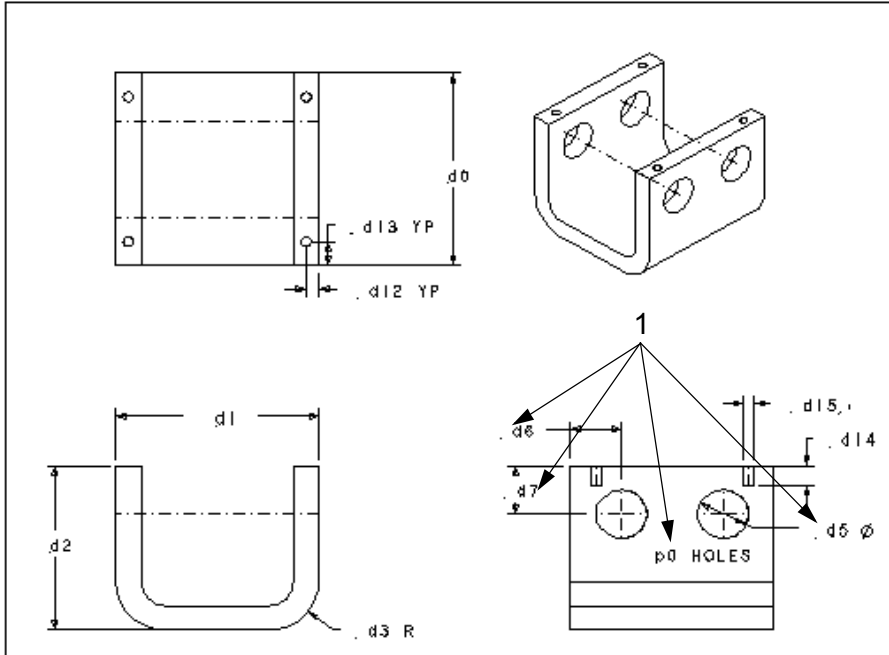
1   Parameter symbols.

Example: A Note with Parameters
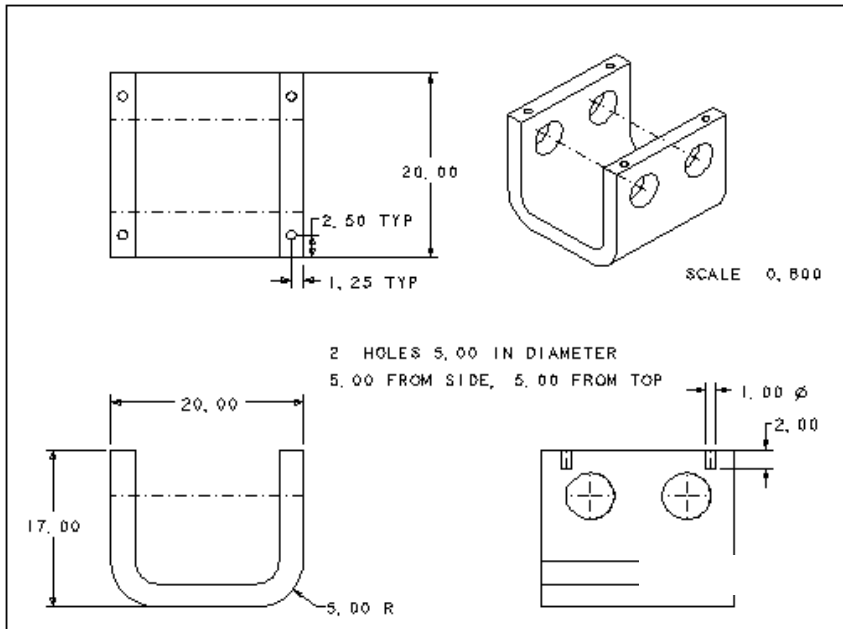
Type the following: &p0 HOLES &d5 IN DIAMETER <ENTER>
                            &d6 FROM SIDE, &d7 FROM TOP <ENTER><ENTER>

To Change a Text String

After you create a note, you can make changes to the text.

1. Click the text to modify.
2. Click and hold the right mouse button until the menu appears.
3. Click **Edit Text** to modify the note text.

Text Strings

When you are editing text, dimension values appear as follows:

- @D (if displayed as a numerical value).
- @S (if displayed as a symbolic value).
- @O (if the dimension does not have a value, @O locates the text origin).
- Special characters appear between the control characters ^A and ^B.

When you edit a note using **Text Line** and **Full Note**, the system preserves all of the attributes (font, height, width, or slant angle) applied to a portion of the text. However, the note appears much different from how it does on the drawing. The system breaks up a text string into portions wherever there is a new line of text or a parameter (such as dimensions), and encloses each portion of the text in braces ({}), giving it an integer label. Labels identify the initial order of the text, and any attributes for that portion. When editing text, or adding more lines, you can copy the attributes of a portion of text by using the same integer label.

Note: If you delete the special control characters when editing text, the text string changes. If you remove ^A, all special characters become their ASCII equivalent; if you remove ^B, all text that follows becomes a special character.

You can edit view-related notes, such as section, detail, and scale notes; however, be sure not to delete or otherwise alter symbols that represent the name or scale of the view. If you edit these symbols, the system ignores the change, and displays the original note:

- `&view_name`—name of the view
- `&view_scale`—name of a general scaled view
- `&det_scale`—scale of a detailed view

If you set the `view_note` option in the drawing setup file to `std_din`, you can create a view-related note with the words `SECTION`, `DETAIL`, and `SEE_DETAIL` omitted. This only affects the creation of view-related notes. If you switch from `std_ansi` to `std_din`, the system does not update view notes.

Guidelines for Using Note Labels

When using note labels, keep in mind the following:

- The system treats text that is not enclosed within braces, or is incorrectly labeled, as plain text.
- Text labeled with a number higher than that which is actually present uses the attributes of the highest number label.
- You can display braces in the text string. For left braces, only include the brace. For right braces, so that the system does not interpret it as the end of a portion of text, type two right braces together (`}}`). For example, for "Origin {X,Y,Z}," enter `[{Origin {X,Y,Z}},}]`.

Example: Effects of Editing Note Text

This note contains dimensions (447.23), fonts
slant angle and height,
and different widths.

Select this note to edit, using **Full Note**.

{0:This note contains dimensions
(}{1:&D2}{2:), fonts}
{3:slant angle and height,}
{4:and different widths.

Note the braces and colons that identify the different portions of text.

{0:This note contains dimensions
(}{1:&D2}{2:), fonts}
{3:slant angle and height,}
{4:and different widths.}
{3:This line uses slant and height
attributes from portion 3}
{0: and this one uses font
attributes from 0}

Modify the note, Notice how existing integer labels were used in the two new lines.

Saving the changes causes the modified note to appear. Notice that lines using the same integer labels appear the same.

This note contains dimensions (447.23), fonts
slant angle and height,
and different widths.
This line uses slant and height attributes from portion 3
and this one uses font attributes from 0

{0:This note contains dimensions
(}{1:&D2}{2:), fonts}
{3:slant angle and height,}
{4:and different widths.}
{5:This line uses slant and height
attributes from portion 3}
{6: and this one uses font
attributes from 0}

Edit the note again. The system automatically renumbered the last two lines to be in sequence with the other portions of text, but this does not alter the style of the lines in any way.

To Specify an Existing Text Style as the Current Style

As you create a note, you can apply an existing text style by choosing **Cur Style** from the NOTE TYPES menu. When you change the current style from the default to another text style, it only affects new notes, balloon text, symbols, and table text.

If you choose **Default** from the SEL STYLE menu, the system uses the style that is defined by the drawing setup file options `drawing_text_height`, `text_width_factor`, and `text_thickness`.

To Specify the Default Font

By default, Pro/ENGINEER creates text in regular ASCII font, referred to in menus as font. A thickened version of the system default font (the filled font) is also automatically available to you as an alternative font. You can specify the ISO font as the default font by setting the drawing setup file option `default_font` to `isofont`, or you can set any of the Pro/ENGINEER-supported fonts as the default for a particular drawing.

In addition to those fonts, Pro/ENGINEER supplies the four auxiliary fonts listed in the following table and 41 True Type fonts.

Leroy font set	`leroy`
Calcomp Alphabetical font set	`cal_alf`
Calcomp Greek and Mathematical Symbol set	`cal_grek`
ISO font set	`isofont`

The TrueType fonts are located in the Pro/ENGINEER loadpoint directory. Additional TrueType fonts can be purchased and placed into the loadpoint directory. TrueType fonts are more complicated than PTC fonts and therefore can take more time to repaint. Typically, any TrueType Font can be used by Pro/ENGINEER by placing it in the `pro_font_dir`. However, PTC does not guarantee the robustness of the TrueType Font. See the Pro/ENGINEER Installation and Administration Guide for information about where to locate the font file in the load directory area.

By default, any fonts placed in the loadpoint directory appear in the pull-down list in the **Text Style** dialog box. A different directory other than the loadpoint directory can be specified using the configuration file option `pro_font_dir`.

The following table lists the available TrueType fonts and their corresponding file names.

TrueTypeFonts

Font Name	Setup Option File Name
Blueprint MT	`bluprnt.ttf`
Blueprint MT Bold	`bluprntb.ttf`
CG Century Schoolbook	`schlbk.ttf`
CG Century Schoolbook Bold	`schlbkb.ttf`
CG Century Schoolbook Bold Italic	`schlbkbi.ttf`
CG Century Schoolbook Italic	`schlbki.ttf`
CG Omega	`cgomg.ttf`
CG Omega Bold	`cgomgb.ttf`
CG Omega Bold Italic	`cgomgbi.ttf`
CG Omega Italic	`cgomgbit.ttf`
CG Times	`cgtime.ttf`
CG Times Bold	`cgtimebd.ttf`
CG Times Bold Italic	`cgtimebi.ttf`
CG Times Italic	`cgtimeit.ttf`
CG Triumvirate	`trium.ttf`
CG Triumvirate Bold	`triumb.ttf`
CG Triumvirate Bold Italic	`triumbi.ttf`
CG Triumvirate Condensed Bold	`triumcb.ttf`
CG Triumvirate Italic	`triumi.ttf`
Garamond	`garamd.ttf`

Garamond Bold	`garamdb.ttf`
Garamond Italic	`garamdi.ttf`
Garamond Kursiv Halbfett	`garamdbi.ttf`
Garth Graphic	`gargra.ttf`
Garth Graphic Black	`gargrabl.ttf`
Garth Graphic Bold Italic	`gargrabi.ttf`
Garth Graphic Italic	`gargrai.ttf`
Grotesque MT	`grotesq.ttf`
Grotesque MT Bold	`grotesqb.ttf`
Microstyle Extended	`microex.ttf`
Microstyle Extended Bold	`microexb.ttf`
Neographik MT	`neograph.ttf`
Sackers English Script	`sackengs.ttf`
Shannon	`shanno.ttf`
Shannon Bold	`shannob.ttf`
Shannon Extra Bold	`shannoeb.ttf`
Shannon Oblique	`shannoo.ttf`
Spartan Four MT	`sparton4.ttf`
Spartan One Two MT	`sparton12.ttf`
Spartan One Two MT Bold	`sparton12b.ttf`
Triumvirate Inserat	`triin.ttf`

If you need additional true type fonts, you can purchase more fonts. For information, contact the Agfa Corporation at http:\\www.agfadirect.com.

Creating Your Own Fonts

You can create your own font and set it as a default or auxiliary font in a drawing. For information on creating user-defined fonts, see the *Pro/ENGINEER Installation and Administration Guide*. Note that when you start Pro/ENGINEER, the system first searches the local directory for the files specified in the index file. If it does not find them, it searches the Pro/ENGINEER loadpoint directory. When you save user-defined fonts locally, you should move them to a new location whenever you transfer the drawing. If the system does not find fonts used in the drawings, it uses a different font and the appearance of the text changes accordingly. Therefore, you should maintain font files in the load directory area.

To Modify Drawing Text Style

1. Click **Format > Text Style**.
2. Select text and choose **GET SELECT > Done Sel**.
3. In the **Text Style** dialog box, change the information in **Character** to modify the text style.
4. Click **Apply**. The text changes. To reset the text to the old style, click **Reset Settings**.

The Text Style Dialog Box

You can change the font, width, and color of an existing text style. However, the system does not automatically update any text to which you previously assigned that text style (that is, before you modified it). The changes that you make using this dialog box only affect text to which you assign a text style after modifying it.

Using the **Text Style** dialog box, you can:

- Change the font by selecting a font name from the **Font** list.
- Change the text by clearing the **Default** check box and typing a value in the **Height** box.

- You can also modify the height by choosing MODIFY TEXT > **Text Height**.
- Change the width by clearing the **Default** check box and typing a value in the **Width Factor** box.
- Change the thickness by clearing the **Default** check box and typing a value in the **Thickness** box.
- Change the slant angle by typing a value in the **Slant Angle** box.
- Underline the text by selecting the **Underline** check box.
- Change the line spacing by clearing the **Default** check box and typing a value in the **Line Spacing** box.
- Rotate the text at an angle by typing a value in the **Angle** box (you can rotate a note about a point by choosing TOOLS > **Rotate**).
- Specify the horizontal justification by selecting a value from the **Justify Horiz** list.
- Specify the vertical justification of table cell text in individual table cells by selecting a value from the **Justify Vert** list.
- Flip the text so that it reads backwards by selecting the **Mirror** check box To mirror a note, you must first select it; then select a mirror line.

Using True Type Fonts

TrueType fonts can be used with Pro/ENGINEER. Pro/ENGINEER supplies 41 TrueType fonts, and you can purchase additional TrueType fonts and place them in the Pro/ENGINEER loadpoint directory. See the Pro/ENGINEER Installation and Administration Guide for information about where to locate the font file in the load directory area. TrueType fonts are more complicated than PTC fonts and therefore can take more time to repaint.

To Modify the Color of Drawing Text Using the System-Supplied Colors

1. Click **Format > Text Style**.
2. Select text; then click **GET SELECT > Done Sel**. The **Text Style** dialog box opens.
3. In the **Text Style** dialog box, select the **Color** box.
4. In the **Color** dialog box, select a system color from the **System Colors** box; then click **OK**. The system displays the selected text in the specified color and closes the Color dialog box.
5. To reset the text to the old style, click **Reset Settings**. To reset selected text to its intrinsic color, click **Intrinsic** in the **Color** dialog box.

To Create Your Own Color

1. In the **Color** dialog box, click **New...**.
2. Using the Pro/ENGINEER Color Editor tool, define the new color by moving the RGB controls from right to left incrementally. Click on the color bar at the desired color setting, or click **OK**. The new color appears in the **User-defined Colors** box.
3. To create a *color.map* file for storing your user-defined text style colors, click **Store Map** in the Color dialog box.
4. Click **OK**. The **Color** dialog box closes.
5. In the Text Style dialog box, click **Apply**. The system displays the selected text in the specified color, and stores the color with the model. To reset the text to the old style, click **Reset Settings**.
   Note: Use the line_style_standard option in the drawing setup file to control text color in Pro/ENGINEER drawings. Unless you set this option to STD_ANSI, all text in drawings appears in blue, and the boundary of detailed views, broken views, and half views appears in yellow.

To Create and Edit Custom Text Styles

Use the **Style Lib** dialog box to create new text styles, as well as modify and delete existing ones.

1. Click **Format > Text Style Gallery**. The **Text Style Library** dialog box opens.
2. Click **New…**. In the New Text Style dialog box, type a name fin the Style Name box, select an existing

style from the **Style Name** list, or click **Select Text…**.

3. Type a name in the **Style Name** box, select an existing style from the **Style Name** list, or click **Select Text...**.
4. After you have changed all attributes, click **OK**.
5. To close the dialog box and save the changes, click **Close** (**Cancel** changes to **Close** after you make a change).

To Modify an Existing Text Style

You can modify the existing text styles of notes and other information in your drawing.

1. Click **Format > Text Style Gallery**. The **Text Style Library** dialog box opens.
2. Scoll through the items in the **Styles** container and select the text style name you want to modify. The Modify Text Style dialog box opens.
3. Click Modify… and change the attributes for the style by making the appropriate selections or typing values (for instructions on how to specify a particular attribute.
4. After you have specified all attributes, click **OK**. The text style changes.
5. To reset the text to the old style, click **Reset Settings**.

To Control the Format of the Date Displayed in a Drawing

The configuration file option `todays_date_note_format` controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order. The default value is `%dd-%Mmm-%yy`.

- Year
  - %yy, for 97
  - %yyyy, for 1997

- Month (if the month contains two digits (for example, 10),% mm, % m, or % m all produce the same result)
  - %Mmm, for Jan
  - %MMM, for JAN
  - %Month, for January
  - %MONTH, for JANUARY
  - %mm, for 01
  - %m, for 1
  - % m, for <space>1

- Date (if 2 digits are needed to represent the date, all three are the same. Therefore, "%dd %mm %yy" produces "01 01 97," and "%MMM %d %yyyy" produces "JAN 1 1997")
  - %dd, for 01
  - %d, for 1
  - % d, for <space>1.

The following formats are also valid:

- %dd-%Mmm-%yy (= 01-Jan-97)
- %mm/%dd/%yy (= 01/01/97)
- %Mmm %dd,%yyyy (= Jan 01, 1997)

To Reference Parameters Assigned to Objects

In model and drawing notes, to reference parameters that you assigned to object s in Part or Assembly mode, use this format:

- For edges:
  ```
  &<param_name>:EID_<edge_name>
  ```
- For surfaces:
  ```
  &<param_name>:SID_<surface_name>
  ```
- For other objects:
  ```
  &<param_name>
  ```

To Include a Feature Parameter in a Note

To include a feature parameter in a note, use this format:

```
&<param_name>:FID_<feat_ID>
or
&<param_name>:FID_<FEAT_NAME>
```

Example: Parameters in Notes

The following example illustrates the parameter's usage in notes:

1. In Part mode, assign the name "A" to a surface.
2. Create a parameter with a name "RAD," of type Number.
3. Assign a value ".03" to this parameter.
4. In Drawing mode, create a note by typing [Break sharp edge with R = &RAD:SID_A]. This note appears: "Break sharp edge with R =.03".

If a drawing note has a single attachment (defined using **On Item** or one leader line), you can show the parameters of the entity to which it is attached by typing the following string into the note: &param_name:att. The system then interprets the param_name in a series of contexts until it is able to evaluate it successfully. First, it searches the immediate entity to which the note is attached, such as an edge. Next, it searches the feature that owns the entity. If it still cannot evaluate the parameter, it then searches the model that owns the feature. Finally, if applicable, it searches the component that refers to the model. If you want to specify the exact context in which to interpret the parameter, specify a full "att_" postfix instead.

After you create the note, if you choose **Switch Dims** from the Pro/ENGINEER **Info** menu, the system appends the "att_" postfix to specify the exact context in which it interprets the parameter.

Param_name:att_edge	Edge
Param_name:att_feat	Feature
Param_name:att_mdl	Model
Param_name:att_cmp	Component

For example, if a note is attached to an edge by a single leader line (or defined using **On Item**), you can set the note to show a relational parameter of that edge by typing [&Length:att]. The note then appears as follows: &Length:att_edge.

Note: You must create the drawing note after you create the parameter; otherwise, the system does not evaluate it correctly.

To Reference Model Notes in Drawing Notes

To reference a model note in a drawing note, do one of the following:

- Create a note in Part or Assembly mode by choosing **Notes** from the SETUP menu.
- Add a note parameter to the model using **Add Param** in the RELATIONS menu; the value for this parameter is a note ID.

To Include a Model Note in a Drawing Note

To include a model note in a drawing note, use the format `&<param_name>`.

Example: Including a Model Note in a Drawing Note

1. In a model, create this note: `all surfaces must be painted` with ID 1.
2. Add a parameter `part_note` with the value of 1.
3. In Drawing mode, type this note:
   `[Applies to all parts:`
   `&part_note]`
   The following note appears:
   Applies to all parts:
   all surfaces must be painted
   Note: You cannot insert the `part_note` text side-by-side with any other note text. When you call out a note parameter in another note, the called out parameter is displayed on a new line.

To Display Pro/PDM Data

You can display the name of a model's product database on a drawing sheet, as well as the model's revision in that product database and its release level, by entering the following in a note:

- To display the model's product database of origin, enter the parameter `&PDMDB`.
- To display the model's revision, enter the parameter `&PDMREV`.
- To display the drawing's revision number, enter the parameter `&PDMREV:D`.
- To display the model's release level, enter the parameter `&PDMRL`.

To Reference a Mass Properties Symbol in a Note

You can create a parametric note that references a mass properties symbol. After the geometry changes, you can update the note to reflect the latest value of the mass properties parameter.

1. Set a user-defined parameter. For example: `[volume]`.
2. Add a relation, assigning this parameter to a mass properties symbol that you want to reference in a note. For example: `volume = mp_volume("")`.
3. Add a note containing the user-defined parameter. For example: `[&volume]`.

To Update a Parametric Note

1. Regenerate the model.
2. Do one of the following:
   - Manually recompute mass properties by choosing **Analysis** > **Model Analysis...**.
   - Include a MASSPROP statement in the program of the model using Pro/PROGRAM.

Controlling the Number of Decimal Places in Parameters

When you append certain parameter symbols with the characters "[.#]", the system displays those parameters with the number of decimal places specified by #, which is an integer. The system rounds the number, but uses the same value. This applies to the following parameters:

- User-defined model parameters or those defined in model relations
- Drawing labels, such as `&scale` (drawing scale) and `&det_scale` (detail view scale)

When adding the note that contains the parameter, append the parameter symbol with "[.#]", where # is the number of decimal spaces to appear. For example, if a detailed view scale is 1.125, and you want to display

only two decimal places, change the drawing label (by using **Text Line** or **Full Note** in the MODIFY TEXT menu) to `&det_scale[.2]`. This displays the scale note as 1.13.

Restrictions with Dimensions and Other Model Parameters

You cannot use this functionality on dimensions or other model parameters, such as pattern parameters. However, if you make a user-defined parameter equal to the model parameter, and then add the user-defined parameter to the note, you can control the display of the model parameter decimal places.

Example: Controlling the Number of Decimal Places in Parameters

For example, for a model parameter "d1" with the value 12.37580:

1. Type the relation as [`length = d1`].
2. Add the drawing note.
3. When specifying the parameter, type [`&length[.3]`]. The parameter appears as 12.376, but the value the system uses in calculations is still 12.37580.

You can also use the **Num Digits** command to modify the number of decimals in a scale.

To Show the Scale of an Individual View

The drawing parameter `scale_of_view_x` evaluates the drawing scale of whichever view has name X. For example, to call out the scale of a view named DETAILED_BAR, type `&scale_of_view_detailed_bar`.

To Add a Jog to a Leader

1. Click **Insert > Jog**.
2. Select a note.
3. Select the leader to which you are adding the jog.
4. Select the location for the jog. The system adds the jog between the first pick and the second pick.

To Delete a Jog from a Leader

When the system adds a jog to a leader the jog remains in the same place on the drawing, regardless of where you move the dimension. To delete the jog, choose **Edit > Delete Jog**, and pick the jog.

Example: Adding Jogs to a Note with Leader
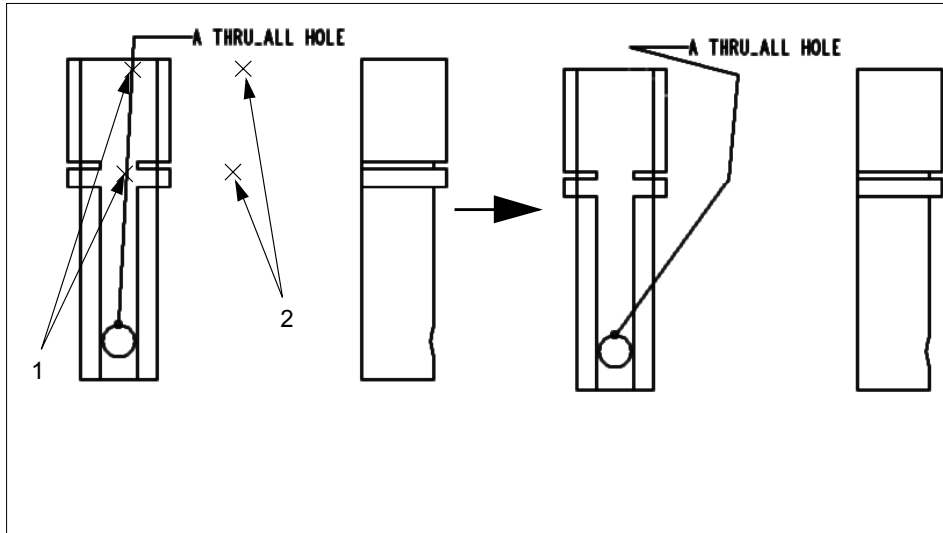
1   Pick the leader at these points.
2   Pick these points as the new position.

To Add Another Leader Line to a Note

1. Click **Insert > Note**. The **Note Type** menu appears.
2. Click **Leader** and then **Make Note**.
3. Select the note to which you want to add the leader.
4. Choose MOD OPTIONS > **Add Ref**.
5. Choose a reference point on the view for the leader.
6. Choose GET SELECT > **Done Sel** and ATTACH TYPE > **Done**. The system adds the additional leader.
   – To add more than one leader, *repeat Steps 4 and 5*.
   – To delete a leader, choose MOD OPTIONS > **Delete Ref**.
   – To break a leader, use the same procedure you would use to break dimension witness lines.
   – Attach a leader to any line of text by typing the placeholder parameter "@o" (alphabetic character, not zero) at the beginning of that line. Add the parameter to the line as you create the note or enter it later using **Full Note** or **Text Line**. If you add "@o" to more than one line of note text, the leader attaches to the first line containing it.

Example: Attaching a Leader to a Note
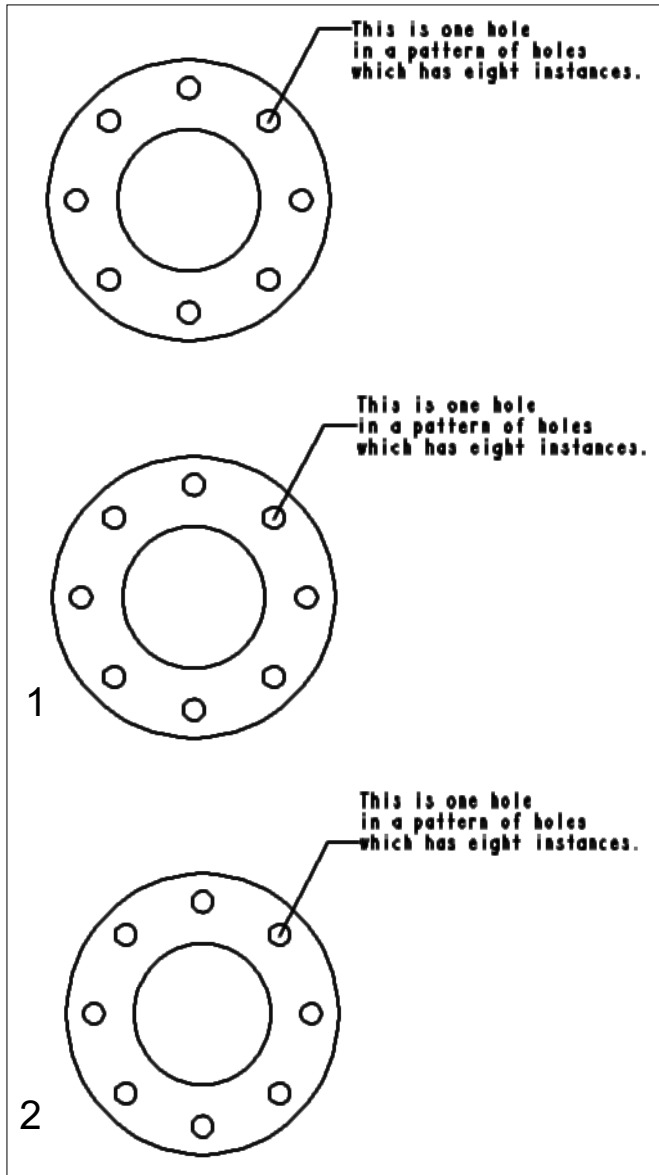
1 Edit the note using **Full Note.** Type "@o" before the beginning of the second line of text.

2 Edit the note using **Full** Note. Remove the"@o" from the second line and add it to the beginning of the third line.

To Delete a Note

Deleting a note permanently removes it from the drawing. The system returns any dimensions or parameters included in that note to their original locations on the drawing. (You can delete notes if you have a license for Pro/DETAIL.)

1. Click **File > Delete** (to remove several notes at one time, use the **Delete Many** button in the Pro/ENGINEER toolbar).
2. Select a note(s).
3. Choose GET SELECT > **Done Sel**. Choose **Repaint**, if necessary, to redisplay the drawing.

To Erase a Note

You can erase notes using the **Show and Erase** command in the menu bar. Erasing a note obscures it from view, but does not remove it from the drawing. The system also hides any dimensions or parameters that are included in the erased note, and does not return them to their original locations elsewhere in the drawing.

1. Choose **View > Show and Erase**.
2. In the **Show and Erase** dialog box, click **Erase** and **Note**.
3. Select a button from the **Erase By** box. The corresponding notes no longer appear. When you have finished, click **Close**.
4. To redisplay erased notes, click **Show** and **Note**.

To Change the Leader Type

If you create a note using the **ISO Leader** command in the NOTE TYPES menu, the system underlines the note. To switch between the ISO-standard leader style and the non-ISO-standard leader style, choose **Leader Type** from the MODIFY TEXT menu and use the GET SELECT menu.

To Enclose Notes in Text Boxes

To enclose words or characters in a box, type "@[" in front and "@]" following the text when you edit, it using the **Text Line** or **Full Note** command.

In a text string, you can place special symbols that indicate a box around text regardless of the location of braces ({ }) that position text breaks.

Example: Enclosing Notes in Boxes

Editing with Text Line	On the screen
{0: notes in drawings}	**notes in drawings**
{0: @[notes@] in drawings}	**notes in drawings**
{0: @[no@]tes in drawings}	**no tes in drawings**

To Change Note Attachment Point

1. Click **Edit > Attachment**.

2. Select the note.

3. Click the new attachment point.

The leader is reattached at the new point.

To Copy a Free Note by Translating

1. Choose TOOLS > **Copy** > **Translate**.
2. Select notes to copy; then choose GET SELECT > **Done Sel**.
3. Define the translation vector.
4. Pick the first point; then select the destination point.
5. Type the number of copies.

To Copy a Free Note by Rotating

1. Choose TOOLS > **Copy** > **Rotate**.
2. Select notes to copy; then choose GET SELECT > **Done Sel**.
3. Select the center point for rotation.
4. Type the rotation angle in the counterclockwise direction.
5. Type the number of copies.
   Note: You can include notes in draft groups.

To Relate an Existing Note to Dimension Text

You can relate an existing note to dimension text—so that it moves with the dimension when the dimension changes location—by using the **Relate Obj** command in the TOOLS menu. When you create a note, you can relate it directly to dimension text by choosing **Dim Related** from the NOTE TYPES menu.

To Switch Notes to Another View

You can switch a note from one view to another view using **Switch to View.** Select the note that you want to switch to another view; then select the new view on which to display the note. The note appears in the corresponding location on that view.

To Show a Thread Note in a Drawing

1. To show thread parameters as a note, you must previously define them in the part by doing one of the following:
   – While creating a thread, supply parameter values through the Pro/TABLE environment.
   – After creating a thread, edit parameters by choosing **Parameter** from the PART SETUP menu.
2. You can move and delete thread notes as regular notes. In Drawing mode, choose **View > Show and Erase**.
3. In the Show/Erase dialog box, click **Show**; then select **Note** from the **Type** box and **Feat & View**.
4. Pick the thread in the drawing view. A note listing thread parameters appears.
5. Show parameter values by choosing **Switch Dims** from the Pro/ENGINEER **Info** menu.

Example: Showing a Thread Note
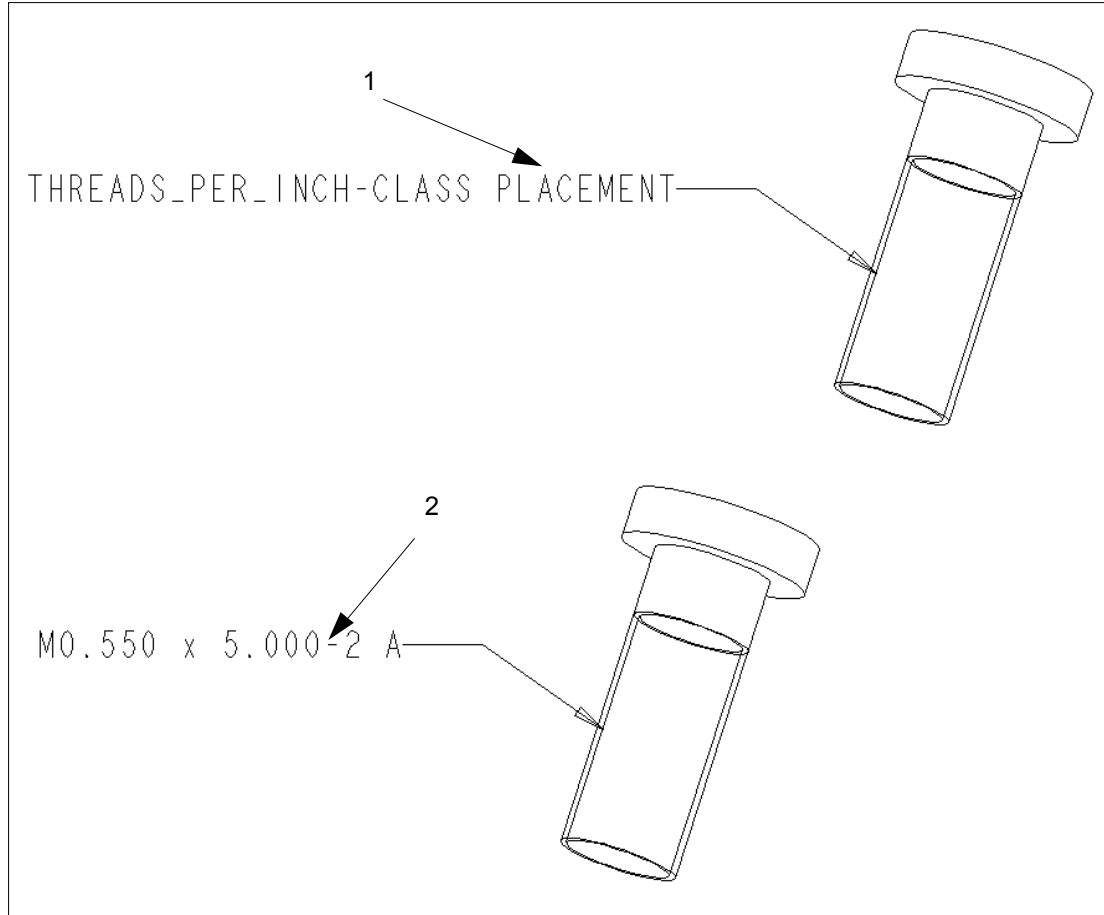
1   When the note appears, it lists parameter names.
2   To show parameter values, switch dimensions.

About Creating Geometric Tolerances

In manufacturing, you can use geometric tolerances to specify the maximum allowable deviation from the exact size and shape specified by designers. You can create, modify, display, and delete geometric tolerances (gtols) in Drawing mode.

Geometric tolerances provide a comprehensive method of specifying where on a part the critical surfaces are, how they relate to one another, and how the part should be inspected to determine if it is acceptable. When you store a Pro/ENGINEER geometric tolerance in a solid model, it contains parametric references to the geometry or feature it controls—its *reference entity*—and parametric references to referenced datums and axes. As a result, the system updates a gtol's display when you rename a referenced datum.

In Assembly mode, you can create a geometric tolerance in a subassembly or a part. A geometric tolerance that you create in Part or Assembly mode automatically belongs to the part or assembly that occupies the window; however, it can refer only to set datums belonging to that model itself, or to components within it. It cannot refer to datums *outside* of its model in some encompassing assembly, unlike assembly-created features.

You can use the geometric tolerance functionality in Drawing mode, Part mode, and Assembly mode; however, to create a geometric tolerance in Drawing mode, you must have a license for Pro/DETAIL.

To Add a Geometric Tolerance to a Drawing

1. Click **Insert > Geometric Tolerance**. The **Geometric Tolerance** dialog box opens.
2. In the **Geometric Tolerance** dialog box, specify the model in which to add the geometric tolerance (the system selects **Model Refs**). By default, the current geometric tolerance model is the current drawing model.
3. Select the geometric tolerance type and the reference entity (the type of entity to which the geometric tolerance applies).
4. Click **Place Gtol**. If the geometric tolerance is attached directly to a datum, it appears (the system selects **Datum** for you from the **Type** list). Otherwise, select an item from the **Type** list and place the geometric tolerance in the drawing.
   Note: As you continue creating the geometric tolerance, the system updates it on the drawing. You can check your work as you go along and make corrections, if necessary.
5. Specify the datum reference(s) and material condition(s), if applicable.
6. Type a tolerance value and material condition, if applicable.
7. Specify symbols and modifiers, the profile direction, and the projected tolerance zone, if applicable. The completed geometric tolerance now appears in the drawing.
8. Do one of the following:
   - Close the dialog box and save the changes by clicking **OK**. The system clears the reference entity selection and placement information from the dialog box, but retains all other data. When you reenter geometric tolerance Creation mode, it retains all commands in the previous session of geometric tolerance creation for the object in the current window.
   - Create another geometric tolerance by clicking **New Gtol**.
   - Delete the geometric tolerance from the model by choosing GEOM TOL > **Clear** to delete the geometric tolerance from the model.
   - Exit geometric tolerance creation mode and cancel the changes by clicking **Cancel**.

Tip: Adding a Geometric Tolerance

To add a geometric tolerance (gtol) to a drawing, you can use two, three, or all four pages of the **Geometric Tolerance** dialog box, depending on the specific characteristics of the geometric tolerance you are creating.

To Place a Geometric Tolerance

You can attach a geometric tolerance to a dimension, datum, single or multiple edges, or another geometric tolerance. You can also create a driven dimension to which you can attach the geometric tolerance, place it free (anywhere on the drawing), or relate it to dimension text. You can attach stacked (multiple) geometric tolerances to another tolerance; or, if the first tolerance in a stack is attached to a dimension, you can attach them to the same dimension. However, to attach a geometric tolerance directly to other geometric tolerances, dimensions, or datums, it must belong to the same model as the item to which it is attached.

Example: Ways of Placing Geometric Tolerances
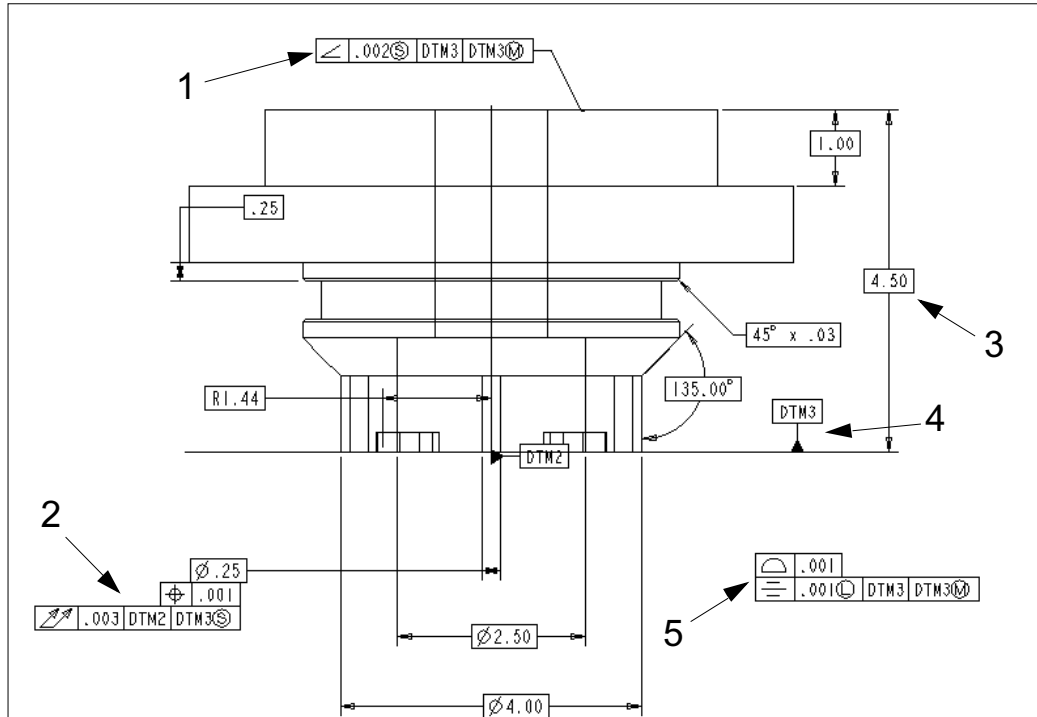
1   Dimension attached to the edge with a leader.
2   Both gtols attached to the dimension.
3   Set dimension.
4   Set datum.
5   First geometric tolerance created as a free note; the second was attached to it.

To Indicate an ISO Standard Projected Tolerance Zone

1. After selecting the **Location/Position** button, specify referenced datums and their material conditions.
2. Click **Symbols** at the top of the dialog box.
3. Do one of the following:
   – To indicate the projected tolerance zone and specify the height, select the **Zone Height** box and type a value.
   – To indicate the projected tolerance zone without specifying the height, placing it on a separate line below the geometric tolerance, select **Below Gtol**.
   – To indicate the projected tolerance zone without specifying the height, placing it on the same line next to the geometric tolerance, select **Inside Gtol**.
   – To specify no projected tolerance zone, select **None**.

The completed geometric tolerance appears with the specified projected tolerance zone. The following figure presents an example.

Example: Projected Tolerance Zone
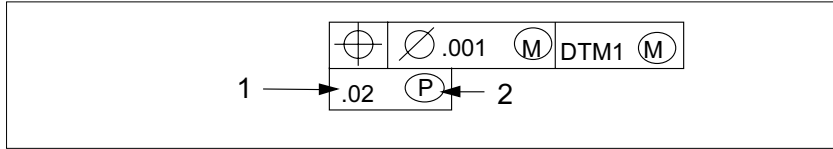
1   The projected tolerance zone height value.
2   The projected tolerance zone height symbol.

To Indicate a Standard Tolerance for Parallelism and Perpendicularity

After selecting the **Orientation/Perpendicularity** or **Orientation/Parallelism** button, specify the reference entity.

1. Click **Tol Value**.
2. Specify the tolerance value by doing one of the following:
    –   To show only the geometric tolerance, without indicating the restricted length, specify a value in the **Overall Tolerance** check box.
    –   To indicate the restricted length to which the tolerance applies, select the **Per Unit Tolerance** check box and type values in the **Value/Unit** and **Unit Length** or **Unit Area** boxes.

Specifying the Per Unit Length Area

To specify the "per unit length or area" of **Form/Straightness** and **Form/Flatness** geometric tolerances, indicate the restricted length to which the tolerance applies by selecting the **Per Unit Tolerance** check box and typing values in the **Value/Unit** and **Unit Length** boxes.

Creating Geometric Tolerances in Assembly Drawings

When you create a geometric tolerance in the top-level model (that is, a part in a part drawing or top assembly in an assembly drawing) the system associates the tolerance with the view in which you have selected a reference entity. Reference datums that you specify for the geometric tolerance *must* belong to the same top-level model; however, you can select them in any view. You *cannot* reference draft entities and draft datums. Geometric tolerances applied to draft entities (draft gtols) can reference datums of the model. You can attach an assembly geometric tolerance to a dimension, datum, or another geometric tolerance, provided they *both* belong to the same assembly.

When you create a geometric tolerance in an assembly drawing, if you set the configuration file option `draw_models_read_only` to yes, the system sets the **Drawing** command as the default in the **Model** list of the **Model Refs** page of the Geometric Tolerance dialog box. The **Part** and **Assembly** commands are not available for selection; if you choose them, an error message appears. If you set `draw_models_read_only` to `no`, and there are no assembly views on the current page, the **Part** command is the default in the **Model** list. If an assembly view is present, **Assembly** is the default.

To Create a Geometric Tolerance in an Assembly Drawing

1. Click **Insert > Geometric Tolerance > Specify Tol**.
2. In the **Geometric Tolerance** dialog box, select **Assembly** from the **Model** list of the **Model Refs** page.
3. Click **Place Gtol**. If the geometric tolerance is attached directly to a datum, it appears (the system selects

Datum for you from the **Type** list). Otherwise, select an item from the **Type** list and place the geometric tolerance in the drawing.
Note: As you continue creating the geometric tolerance, the system updates it on the drawing. You can check your work as you go along and make corrections, if necessary.

4. Specify the datum reference(s) and material condition(s), if applicable.
5. Type a tolerance value and material condition, if applicable.
6. Specify symbols and modifiers, the profile direction, and the projected tolerance zone, if applicable. The completed geometric tolerance now appears in the drawing.
7. Do one of the following:
    -   Close the dialog box and save the changes by clicking **OK**. The system clears the reference entity selection and placement information from the dialog box, but retains all other data. When you reenter geometric tolerance Creation mode, it retains all commands in the previous session of geometric tolerance creation for the object in the current window.
    -   Create another geometric tolerance by clicking **New Gtol**.
    -   Delete the geometric tolerance from the model by choosing GEOM TOL > **Clear** to delete the geometric tolerance from the model.
    -   Exit geometric tolerance creation mode and cancel the changes by clicking **Cancel**.

Example: Adding a Geometric Tolerance to a Drawing

Following the procedure below, use the characteristics and values given in this table to create a sample geometric tolerance.

Sample geometric tolerance Characteristics and Values

Characteristic	Value
Entity	Hole
Tolerance Location	As part of a diameter dimension
Class and Type	Location/Position (a "true position" tolerance)
Overall Tolerance Value	0.25
Material Condition	MMC
Primary Datum (A) Material Condition	RFS/Default
Secondary Datum (B) Material Condition	RFS/Default

1. Click **Insert > Geometric Tolerance**.
2. In the **Geometric Tolerance** dialog box, specify the model in which to add the geometric tolerance (the system selects **Model Refs**). By default, the current geometric tolerance model is the current drawing model.
3. Click **Location/Position**.
4. Specify the type of entity to which the tolerance applies. Select **Feature** from the **Reference Entity Type** list; then click **Select Entity** to select the hole in the drawing.
5. Click **Place geometric tolerance** and select **Dimension** from the **Type** list. Select the hole's dimension. The geometric tolerance appears in the drawing below the dimension.
6. Click **Datum Refs** at the top of the dialog box:
    -   To specify the primary datum reference, click **Primary**; then select **A** and **RFS (no symbol)** from the **Basic** lists.
    -   To specify the secondary datum reference, click **Secondary**; then select **B** and **RFS (no symbol)** from the **Basic** lists.
7. Click **Tol Value** at the top of the dialog box and type [0.25] in the **Overall Tolerance** box. Select **MMC** from the **Material Condition** list.
8. Click **Symbols** at the top of the dialog box. Select the **Diameter Symbol** check box to indicate that the tolerance value refers to a diameter zone; then select **None** as the projected tolerance zone.

9. The completed geometric tolerance appears, as shown in the following figure. Click **OK** to close the dialog box and save the changes.

Sample Gtol
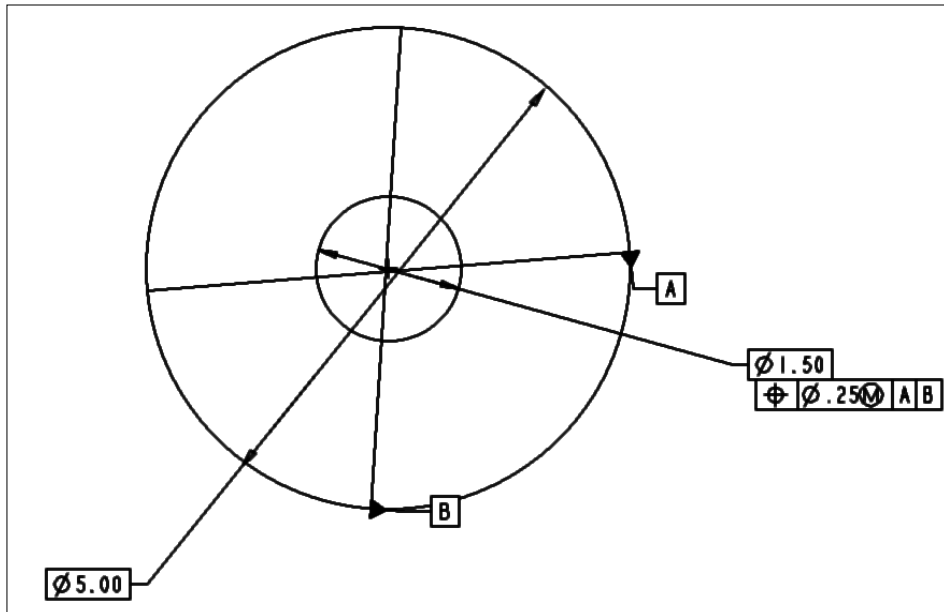
About Modifying the Dimensioning Scheme

You can modify the dimensioning scheme of a feature in Drawing mode by using **Edit > Dimension Scheme** in the menu bar. When you select a feature, a subwindow opens and displays the model. You can then modify the section of the feature using commands in the SKETCHER menu.

To Modify the Dimensioning Scheme of Feature or Part

1. In the menu bar, click **Edit > Dimension Scheme**.
2. Select a feature to redimension. If you selected the base feature, only the section of the base feature and dimension values appear in a separate window. If you selected a later feature on the model, the system rolls back the model and displays it in default orientation in a separate window. The section and dimensions of the feature (if applicable) appear on the model.
3. In the dialog box that displays the elements in the selected feature, add dimensions, delete dimensions, or change placement constraints, as necessary. You *cannot* modify existing dimension values or add or delete features.
4. Click **OK** when you are done.
5. The system regenerates the model to reflect the changes. If you set the configuration file option `auto_regen_views` to `no`, you must choose **View > Update > Current Sheet** or **All Sheets** to observe the change in the drawing views.

   You can also modify the dimensioning scheme in Part mode.

About Using Snap Lines in Drawing Views

You can define snap lines on individual drawings to locate dimensions, notes, geometric tolerances, symbols, and surface finishes. The system positions the snap lines relative to the view outline, or a selected model edge or datum plane. After you have placed an item on a snap line, the item moves along if the grid line moves (for example, when the view outline expands).

When placing and locating items on a snap line, keep in mind the following:

- When you move an item onto one snap line, its color changes to magenta. If you set the location by pressing the left mouse button, the item *snaps* to the snap line. Until you move the item again, the snap line determines its location.
- If you move an item onto the intersection of two snap lines, the system highlights one of the lines in red. If it snaps that item to more than one set of snap lines at that location, you can navigate all possible sets using the SEL SNAP LINE menu. When you choose **Accept**, the system locates the item on the intersection of the two snap lines. When you move either snap line, the item moves with it.

To Create a Snap Line

1. On the menu bar, click **Insert > Snap Line**.
2. On the Menu Manager, do one of the following:
   - Choose CR SN LINE > **Att View**. Select a view border, specify the offset from the border and the number of lines to create. If you are creating more than one line, specify the space between the two lines. The snap line attaches to the specified view border.
   - Choose CR SN LINE > **Att Geom/Snap**. Select view geometry (such as an edge), a datum plane, or another snap line; specify values for the offset, the number of lines to create, and the space between the two lines (if you are creating more than one line). The snap lines attach to the specified view geometry, datum plane, or snap line.

Restrictions When Using Snap Lines

When working with snap lines, keep in mind the following:

- Snap lines place and locate detail items. They are *not* a graphical part of the drawing, so the system does not plot them.
- You can put snap lines on drawing layers and blank them, but once you blank them, you cannot add new items to them. Existing items continue to snap.
- You cannot add entities of one view to another view's snap line. In the case of broken views, you cannot snap an item of one segment of the view to a snap line of another segment. The view of an item is defined as the view to which the leader of the item is attached. In the case of multiple leaders, the item can snap to any one of the views.
- When you delete a view, the system also deletes all of its snap lines.
- If you place a dimension on an intersection of two snap lines, it does not behave the same way it would if it was placed on only one snap line. Free items can snap to any snap lines.

To Place and Locate Items on a Snap Line

You can snap an item to one snap line or to the intersection of two snap lines. You can also attach clipped detail entities (such as dimensions, witness line endpoints, set datum endpoints, and axis endpoints). Once you place an item on a snap line, it moves when the snap line moves.

1. To place and locate the following items on snap lines, select the item(s) and move them to the snap line. An item "snaps" to the line as you move it closer, and the line and entity colors change to magenta. You can place the following items on snap lines:
- Dimensions
- Clipped dimension arrows
- Notes
- Symbols
- Set datum names
- Set datum line endpoints
- Geometric tolerances

- Surface finishes
- View arrows

To Modify a Snap Line Attachment

1. Select a snap line, and then right-click the line to display the shortcut menu.
2. From the shortcut menu, choose **Mod Attach**. The MOD SNAP ATT menu opens in the Menu Manager. Choose one of these commands:
   - **Keep Position**—Maintains the position of the snap line.
   - **Change Orient**—Reroutes the snap line, changing its orientation but assigning the offset of the existing reference to the new reference.
   - **Att View**—Reroutes the snap line attachment to a view border.
   - **Att Geom/Snap**—Reroutes the snap line attachment to other geometry or another snap line.
3. Specify a new reference. The snap line attachment changes.

You can manipulate snap lines in several ways:

- To remove a snap line, select the line and then click **Edit > Delete**, or select and right-click the line and then choose **Delete** from the shortcut menu. To select more than one snap line at once, hold down the SHIFT key while selecting each line. Once you delete a snap line, any items located on it are free.
- To change the position of a snap line, select the line and then drag it to its new position. The system updates all items located on the moved snap line after you choose a new location.
- To control the length (position of the endpoints) of a snap line, select the line and then click an endpoint (handle). Move the pointer to the new location; the line lengthens or shortens as you move the pointer. Click again to place the endpoint (handle) in the new location.

   Note: Any modifications that you make are cosmetic and affect only the snap lines. You cannot change the length of more than one snap line at a time.
- To change the spacing of snap lines, select the line, right-click on the line, and then choose **Modify Space** from the shortcut menu. Enter a new spacing value and then press ENTER. The system updates all items located on the moved snap line after you choose a new location.

To Control the Display of Snap Lines

1. Click **Utilities > Environment**. The **Environment** dialog box opens.
2. Under **Display**, select **Snap Lines**.

To turn on snapping:

Under **Default Actions**, select **Snap to Snap Lines**. By default, the system selects both commands. If snap lines do not appear or you have cleared the **Snap to Snap Lines** checkbox, the items that are already snapped to a snap line continue to move with their snap lines.

About Working with Drawing Parameters

Using MODEL PARAMS menu (DRAWING > **Advanced > Parameters**), you can access drawing parameter functionality for drawings and drawing formats. Drawing parameters work in the same way as do model parameters. A *drawing parameter* is nongraphical information you can add to a drawing. It is useful for keeping some additional information with a drawing that you may not want to include in a note. You can show one in a note by including [`&parameter:d`] in the note string.

When working with drawing parameters, keep in mind the following:

- The system associates parameters only with the object that is current at the time you add the parameter.
- Drawing attributes defined in versions of Pro/ENGINEER prior to Release 15.0 convert to drawing parameters automatically.
- The system converts attribute names that begin with a number, but you cannot use them in notes.
- You cannot use drawing parameters in relations.

- You can designate parameters for Pro/PDM.

To Create a Drawing Parameter

1. Choose ADV DWG OPTS > **Parameters** > **Drawing** > **Create**.
2. From the ADD PARAM menu, choose a command to specify the parameter type.
   IntNumber-Adds a parameter in the form of an integer.
   RealNumber-Adds a double parameter.
   String-Adds a parameter in the form of a string.
   Yes No-Adds a parameter with a value of `yes` or `no`.
3. Type the parameter name and the value.

To Modify or Delete an Existing Drawing Parameter

1. From the MODEL PARAMS menu, choose **Modify** or **Delete**.
2. From the PARAMETER menu, select a drawing parameter.
   - To modify the parameter, type the new value.
   - To delete it, choose **Done**.
3. To display an Information window showing all of the existing parameters and their values, choose MODEL PARAMS > **Info**.

Note: You also have access to Part and Assembly mode parameters while in Drawing mode. Once you delete a parameter, you cannot undo the deletion.

To Get Information About Drawing Parameters

1. Click DRAWING > **Advanced** > **Parameters** > **Drawing**.
2. In the MODEL PARAMS menu, click **Info**.

An information window opens and lists information about all parameters in the drawing.

To Save Drawing Parameter Information as a File

1. Choose DRAWING > **Advanced** > **Parameters** > **Drawing** > **Info**. An information window opens and lists information about all parameters in the drawing.
2. Click **File > Save As** in the Information window.
3. In the **Save As** dialog box, navigate to the directory where you want to save the parameter information file.
4. In the **New Name** box, use the default file name or type a new name. Parameter information files always contain the extension `.inf`.
5. Click **OK** to save the information file.

About Creating and Modifying Line Styles

A line style consists of the following elements: line font (a pattern of dashes, spaces, and/or dots), color, and line width. Modifying a line style involves changing all or any of the line style elements. You can create, modify, and delete line styles and line fonts of table grids, symbols, axes, draft entities, and cosmetic features.

When you are in Part mode, user-defined line fonts appear in solid font.

To Access Line Styles in Drawing Mode

When you are in Part mode, user-defined line fonts appear in solid font. To access the line style functionality in Drawing mode, click **Format > Line Style** on the Pro/ENGINEER menu bar and choose options as necessary in the LINE STYLES menu (Menu Manager).

To Specify the Default Line Style Setting

1. On the menu bar, click **Format > Default Line Style**. The SEL STYLE menu opens in the Menu Manager.
2. From the SEL STYLE menu, choose one of these commands:
   - **Hidden**—Displays as hidden line (gray) geometry on the screen, and plots as dashed lines.
   - **Geometry**—Displays as regular visible geometry (white) on the screen, and plots as solid lines.
   - **Leader**—Displays as dimensions (yellow) on the screen, and plots as yellow lines.
   - **Cut Plane**—Displays as white phantom lines on the screen, and plots as phantom lines with Pen 1.
   - **Phantom**—Displays as gray phantom lines on the screen, and plots as phantom lines with Pen 3.
   - **Centerline**—Displays as yellow centerlines on the screen, and plots as centerlines.

To Assign a Line Style to Objects

Using **Modify Line** in the LINE STYLES menu, you can access the Line Style dialog box to assign attributes to selected lines.

1. On the menu bar, click **Format > Line Style**.
2. From the LINE STYLES menu in the Menu Manager, choose **Modify Lines**.
3. Select the line you want to modify, and then choose **Done Sel** from the GET SELECT menu. The **Modify Line Style** dialog box opens.
   Note: In order to access this dialog box, you must choose **Done Sel** from the GET SELECT menu after selecting lines.
- To assign a *line style* to entities, select a line style from the **Style** list; then click **Apply**.
- To set the *line font*, select a line font from the **Line Font** list; then click **Apply**.
- To set the *width*, type a value in the **Width** box; then click **Apply** (this text box is only available in Drawing mode; it does not apply when modifying the line style of part entities in any other mode).

Accessing Line Style in Part Mode

In Part mode, you can access the Line Style dialog box by choosing **Line Style** from the MODIFY menu. It contains a subset of the controls used in Drawing mode.

To Set the Color Using a System-Supplied Color

1. On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager.
2. Choose **Modify Lines** from the LINE STYLES menu.
3. Select the item to modify; then choose GET SELECT > **Done Sel**.
4. In the **Modify Line Style** dialog box, click **Color**.
5. In the **Color** dialog box, select a system color from the **System Colors** box; then click **OK**. The system displays the selected item with the specified color and closes the **Color** dialog box.
6. To reset the item to the old style using the old colors, click **Reset** and **Apply**.

To Create Your Own Line Color

1. On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager.
2. Choose **Modify Lines** from the LINE STYLES menu.
3. Select the item to modify; then choose GET SELECT > **Done Sel**.
4. In the **Modify Line Style** dialog box, click **Color**.
5. In **the Color** dialog box, click **New**.
6. In the Pro/ENGINEER Color Editor tool, using the left mouse button, define the new color. Move the RGB **controls** from right to left incrementally; then click on the color bar (or click **OK**) at the desired setting. The new color appears in the **User-defined Colors** box.
7. Click **Apply in the Modify Line Style dialog box. The system displays the selected item with the specified color and** stores the color with the model.

8. To reset the item to the old style, click **Reset** and **Apply**.

To Create (or Add) a New Line Style

1. Do one of the following:
   - On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager. Then, choose **Edit Styles** from the LINE STYLES menu.
   - On the menu bar, click **Format > Line Style Gallery**.

   In either case, the **Line Style Library** dialog box opens.
2. In the **Line Style Library** dialog box, click **New**.
3. In the **New Line Style** dialog box, type a name in the **New Name** box, select an existing style to copy from the **Style** list, or click **Select Line**.
4. Set the line font, width, and color.
5. To accept the new line style as currently defined and change the contents of the Line Style Library, click **OK**. The system adds the line style to the Line Style Library.

To Delete a User-Defined Line Style

1. Do one of the following:
   - On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager. Then, choose **Edit Styles** from the LINE STYLES menu.
   - On the menu bar, click **Format > Line Style Gallery**.

   In either case, the **Line Style Library** dialog box opens.
2. In the **Line Style Library** dialog box, select an existing line style from the scrollable **Styles** container.
3. Click **Delete**.
4. In the Confirm Deletion menu, click **Yes**. The system removes the line style.

To Modify a User-Defined Line Style

1. Do one of the following:
   - On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager. Then, choose **Edit Styles** from the LINE STYLES menu.
   - On the menu bar, click **Format > Line Style Gallery**.

   In either case, the **Line Style Library** dialog box opens.
2. In the **Line Style Library** dialog box, select an existing line style from the **Styles** list.
3. Click **Modify**.
4. In the **Modify Line Style** dialog box, modify the selected line style, as necessary:
   - To change the *font*, select a font from the **Line Font** list.
   - To change the *width*, type a value in the **Width** box.
   - To change the *color*, click **Color** and select a color from the Color dialog box.

   Note: The system does not automatically update an item that was previously assigned that line style (before you modified it). The changes that you make using this dialog box only affect items to which you assign the user-defined line style after having modified them.

About Creating and Modifying Line Fonts

Using the **Edit Fonts** command in the LINE STYLES menu, you can create your own line fonts by specifying the font length and the font pattern, and you can modify existing fonts. A line font is a pattern of dashes and spaces that are proportionate within the specified length. For example, 5 dashes and 5 spaces with a pattern length of 1 inch results in two dashes and spaces for a 2-inch line in the drawing. The same pattern definition with a 1/4-inch pattern length results in 8 dashes and spaces for a 2-inch line in the drawing. You can have a maximum of 16 dash-space combinations in a line font file.

To Create a New Line Font

1. Do one of the following:
   - On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager. Then, choose **Edit Fonts** from the LINE STYLES menu.
   - On the menu bar, click **Format > Line Font Gallery**.

   In either case, the **Line Font Library** dialog box opens.
2. In the **Line Font Library** dialog box, click **New**.
3. In the **New Line Font** dialog box, type a name in the **New Name** box, select an existing font to copy from the **Font** list, or click **Select Line**.
4. Specify the font length by typing the length of one unit in the **Unit Length** box.
5. Specify the font pattern by typing a series of dashes (-) and spaces ( ) in the **Font Pattern** box.
6. Click **OK** to accept the font as currently defined. The system adds the font to the Line Font Library.

Accessing Line Font Files

User-defined line font files, "*fontname.lsl*," must reside in your local directory for you to be able to use them. When you retrieve a drawing, and the system cannot find a line font file, it displays any line that has been set with that style as solid and notifies you that it has not loaded a font file. If you have specified auxiliary font files in the drawing setup file (this occurs automatically when you create the font when that drawing is active), it identifies the font files that it could not find. To use line fonts in drawings other than those in which they were created, set the drawing setup file option `aux_line_font` to specify the line font names and IDs:

    aux_line_font # font filename
where "#" is an integer number between 1 and 10,000 representing the font ID in the drawing. It serves as a cross-reference between geometry and the line font, enabling you to make blanket changes to fonted geometry by modifying the drawing setup file and specifying a new font name for the same number. For example, changing the value of `aux_line_font` from "100 dash 1" to "100 solidfont" changes all geometry that was originally "dash-1" to "solidfont."

To Set the Default Length of a Font

To set the default length of a font, use the drawing setup file option `line_style_length`.

1. Click DRAWING > **Advanced > Draw Setup**. The **Options** dialog box opens.
2. Add the option `line_style_length` to the drawing setup file.

After you add this option to the drawing setup file, you *cannot* delete it by deleting the row from the file or by retrieving a different ".dtl" file into the drawing. You *must* change the value of this option to "default" to eliminate the option from the drawing setup file. Use the following format:

    line_style_length font_name value/default
where `font_name` is the name of the font that you want to modify, `value` is the desired value for the font length in system units, and `default` tells the system to use the default length value. You *must* specify this option in the drawing setup file whenever you want to modify the length. To set the length of sketched lines using `line_style_length`, you must set the drawing setup file option `axis_interior_clipping` to no. *The drawing setup file option "line_style_length" only affects sketched entities.*

Note: Any modifications that you make to the length of a font in a drawing using the **Modify Line Font** dialog box *override* the default setting that you specify in the drawing setup file.

To Modify a Line Font

1. Do one of the following:
   - On the menu bar, click **Format > Line Style**. The LINE STYLES menu appears in the Menu Manager. Then, choose **Edit Fonts** from the LINE STYLES menu.
   - On the menu bar, click **Format > Line Font Gallery**.

   In either case, the **Line Font Library** dialog box opens.

Pro/DETAIL

2. In the **Line Font Library** dialog box, select an existing font from the scrollable **Fonts** container.
3. Click **Modify**.
4. In the **Modify Line Font** dialog box, make the necessary modifications to the selected font:
   – Change the length by typing a new value in the **Unit Length** box.
   – Change the font pattern by typing a series of dashes (-) and spaces ( ) in the **Font Pattern** box.
5. Click **OK** to accept the modifications and change the contents of the font library. The Modify Line Font dialog box closes.

To Delete a Line Font

To delete a line font from the drawing, change the value of the drawing setup file option `aux_line_font` to `<number> solidfont`.

To Import and Export User-Defined Fonts

In Drawing mode, the configuration file option `iges_out_dwg_line_font` controls the export of user-defined line fonts through IGES, while `iges_in_dwg_line_font` controls importing. Pro/ENGINEER processes the IGES line font definition entity (type 304, form 2) when you set these options to `yes`.

About Using Model and Draft Grids

When you are working with a drawing, you can use a three-dimensional (model) grid that was defined in Part or Assembly mode or a two-dimensional grid. To use model grids, click **View > Model Grid**, and then use the **Model Grids** dialog box. To use draft grids, click **View > Draft Grid**, and then use the GRID MODIFY menu in the Menu Manager.

To Create or Modify a Model Grid

You can create a model grid in Part or Assembly mode by choosing **Grid** from the SET UP menu and then selecting a coordinate system to define the origin of the grid from the **Model Grids** dialog box. In an assembly, this coordinate system must belong to the top-level assembly.

To create a model grid in Drawing mode:

1. On the menu bar, click **View > Model Grid**. The **Model Grids** dialog box opens.
2. Use the various options on the **Grids** and **Settings** tabs to create the model grid.
To modify the model grid, follow the same procedure.

To Delete a Model Grid from a Part or Assembly

Note: You can only delete a grid when you are in Part or Assembly mode.

1. In Part or Assembly mode, click **Set Up > Grid** on the Menu Manager. The **Model Grids** dialog box opens.
2. Click **Delete Grid**. The grid disappears.

The Model Grids Dialog Box

Create and modify model grids by using the **Model Grids** dialog box. To access the dialog box in Drawing mode, click **View > Model Grid** on the menu bar. In Part or Assembly mode, click **Set Up > Grids** in the Menu Manager.

The **Grids** tabbed page contains the following options:

- **Origin**—Allows you to select the coordinate system for the grid origin.
- **Display**—Allows control over the grid display.
   – **Show**—Show the grid and grid labels by view, line, or sheet.

- **Erase**—Erase the grid and grid labels by view, line, or sheet.
- **Show/Erase by**:
  - **View**—Show or erase the grid in selected views.
  - **Line**—Show or erase individual grid lines.
  - **Sheet**—Show or erase the grid over the entire sheet. All relevant views snap to the grid.
- **Spacing**—Allows you to assign the same spacing to the entire model grid or individually by axis.
  - **All**—Spaces all the grid lines at the distance you define in the text box.
  - **By Axis**—Spaces the grid lines on any axis individually, at the distances you define in the text box.
  - **View**—Applies modifications to the model grid after you display the model grid.
- **Delete Grid**—Deletes the grid in the current view.

The **Settings** tabbed page contains the following options:

- **Setup Options**—Allows you to assign various values to the following:
  - **Balloon Radius**— Type in the balloon radius.
  - **Offset Distance**—Type in the offset distance of the grid lines.
  - **Decimal Places**—Type in the number of decimal places.
  - **Negative Prefix**—Type in the symbol for the negative prefix to be displayed in balloons. The default sign is "-".
  - **Text Orientation**—Assign the text to be horizontal or parallel to the grid line.
  - **Text Position**—Assign the text to be centered, above the grid line or below the grid line.
  - **Display Balloon**—Allows you to display a balloon.
- **Label Text**
  - **Show Additional Text**—Select this check box if you want to type additional text as a prefix or suffix on any axis.

Considerations When Using the 3-D Model Grid

When using the 3-D model grid, keep in mind the following:

- If you reorient a view with a grid, the system may erase the grid from the display. Therefore, before completing the procedure, you must confirm that you want to reorient the view.
- When you change the view scale, the model grid adjusts to cover the entire view.
- The system extends the grid lines beyond a drawing view outline at a distance that equals two default grid spaces. It uses the default grid when the model grid first appears. You can modify it later, but the extension distance does not change.

To Display a Model Grid in a Drawing

When in a drawing, you can use commands in the DWG MDL GRID menu to display and erase the model grid. To display it, you must orient the views so that one axis of the coordinate system is normal to the screen and one axis is parallel to a drawing sheet border.

1. Choose **View > Model Grid > Show Grid**. The **Model Grids** dialog box opens.
2. Using the **Grids** and **Settings** tabs, specify the display.
   Note: If you add a view to a sheet that has a sheet grid, the view's model has a defined grid origin, *and* the view is in planar orientation to this grid, the system moves the view origin to the grid origin and snaps that point to the grid point of the sheet.

Example: Model Grid
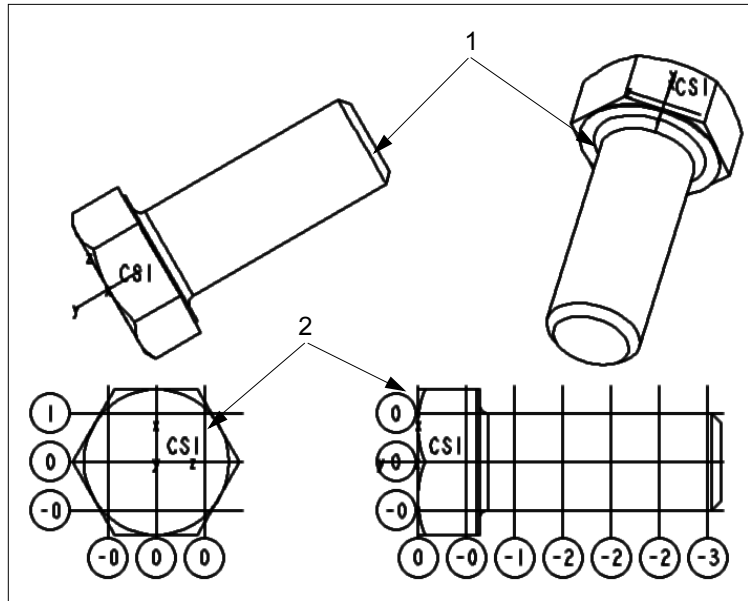
1   Invalid views for model grid.
2   Model grid shown by view.

To Erase a Model Grid from a Drawing

1. On the menu bar, click **View > Model Grid**. The **Model Grids** dialog box opens
2. Do one of the following:
    – To erase the grid by view, click **View** and then select the view. Click **Erase**. The system erases the grid but does not permanently remove it from the model.
    – If you want to erase a grid line, click **Line** and then select individual lines of the grid. Click **Erase**.
    – If you want to erase by sheet, click **Sheet**. The system will ask you if you want to erase the grid on the current page. Accept the default response yes.
3. To redisplay individually erased lines, click **Show > View**.

Modifying Model Grid Size

You can set the default model grid spacing by specifying the value for the configuration file option model_grid_spacing in drawing units. You can also use the **Spacing** options in the **Model Grids** dialog box (**View > Model Grid**).

To Modify the Grid Size

1. On the menu bar, click **View > Model Grid**. The **Model Grids** dialog box opens.
2. If the system displays the grid on individual views, select the view that you want to change.
3. Under **Spacing**, Type a new spacing for the grid. The system updates the grid automatically in all views in which it appears.

To Show Model Grid Balloons

You can show model grid balloons by using the **Model Grids** dialog box or by using the configuration options model_grid_balloon_size, model_grid_neg_prefix, model_grid_num_dig_display, and

`model_grid_offset`.

You access the **Model Grids** dialog box by clicking **View > Model Grid**.

- To specify the default grid balloon size, use **Setup Options** in the **Model Grids** dialog box, or set `model_grid_balloon_size`.
- To control the prefix of negative values in grid balloons, use the **Negative Prefix** option in the **Model Grids** dialog box, or set `model_grid_neg_prefix`. The default value is "-."
- To set the number of digits displayed in grid balloons, use the **Decimal Places** option in the **Model Grids** dialog box, or set `model_grid_num_dig_display`.

To specify the offset of new model grid balloons from the drawing view, set `model_grid_offset`. If you set it to `default` (the default value), the system offsets new model grid balloons from the drawing view by twice the current model grid spacing. If you specify a number as the value, the system offsets balloons by that number of inches (not drawing units) from the view.

To Erase Balloons

When you erase a balloon, you blank it from the display. The balloon still exists in the background. You can redisplay the balloon using the **Show/Erase** dialog box, accessed by clicking **View > Show and Erase** in the menu bar.

1. Select the balloon(s) you want to erase, and then right-click anywhere in the drawing window to display the shortcut menu.
2. Click **Erase**. The selected balloons display in grey, indicating that they have been erased from the display.

Note: If you have erased a balloon in error, you can undo the erase operation immediately if you have not yet clicked another place in the drawing window. While the greyed-out version of the balloon still exists on the screen, right-click and select **Unerase** from the shortcut menu. The balloon becomes highlighted in magenta; if you click anywhere else in the drawing window, the balloon then displays in the normal system color. The **Unerase** option is not available if you have repainted or regenerated the display, or if you have clicked another place in the drawing window after erasing the balloon.

About Creating a Draft Grid

Using commands in the GRID MODIFY menu (accessed by clicking **View > Draft Grid** on the menu bar), you can create two types of draft grids in a drawing: Cartesian and Polar. The system creates a *Cartesian* grid by locating points on a plane, measuring their distance from either of two intersecting straight-line axes along a line parallel to the other axis. It creates a *Polar* grid by locating points in a plane, measuring their distance from a fixed point on a line and the angle this line makes with a fixed line. To change from one type to another, choose **Type** from the GRID MODIFY menu.

When using a two-dimensional draft grid in a drawing, keep in mind the following.

- The grid snap falls on the grid lines when grid snap is on (set in the **Environment** dialog box).
- The grid origin and angle affect the coordinate values specified for geometry points. The x-axis is always along one direction of the grid, while the y-axis is along the other.
- The grid origin affects the coordinate values appearing in the message area when sketching.
- If you use the GET SELECT menu to select an entity, the entity does not snap to grid.

To Change the Grid Display

1. Click **View > Draft Grid**. The GRID MODIFY menu appears in the Menu Manager.
2. Use the **Show Grid** and **Hide Grid** commands in the GRID MODIFY menu.

Note: Turning on the display of the grid does not affect the snapping of sketched entities to grid intersections. To change to grid snap, select **Snap to Grid** from the **Environment** dialog box.

Places Where you can Locate the Grid Origin

You can locate the grid origin at one of the following places. Use **View > Draft Grid** on the menu bar, then click **Origin** on the GRID MODIFY menu.

- Sketched entity endpoint or center
- Sketched point
- Datum point
- Edge or curve vertex

To Move the Grid Origin

1. On the menu bar, click **View > Draft Grid**. The GRID MODIFY menu opens in the Menu Manager.
2. Choose GRID MODIFY > **Origin**.
3. From the GRID ORIGIN menu, choose the type of entity to set the origin:
   - **Get Point**—Selects a point to be the origin.
   - **Edge/Entity**—Selects a point on an edge or entity.
   - **Default**—Uses the default grid origin.
4. If you choose **Get Point** or **Edge/Entity**, select the entity to locate the origin of the grid.

To Modify Grid Spacing

Using the following procedure, you can modify the spacing and angle of the grid lines. The commands that are available depend upon the type of grid. The system saves the values for the grid spacing and retrieves them with the drawing.

1. On the menu bar, click **View > Draft Grid**. The GRID MODIFY menu appears in the Menu Manager.
2. Choose **Grid Params** from the GRID MODIFY menu. For a Cartesian grid, the CART PARAMS menu displays. For a Polar grid, the POLAR PARAMS menu displays.
3. Modify the grid spacing according to the menu that displays as mentioned in the preceding step.

The CART PARAMS Menu and the POLAR PARAMS Menu

Depending on whether the draft grid in a drawing is Cartesian or Polar, the CART PARAMS or POLAR PARAMS menu is displayed when you click **Grid Params** on the GRID MODIFY menu (**View > Draft Grid**). Cartesian grid (and thus the CART PARAMS menu) is the default draft grid.

For a Cartesian grid, the CART PARAMS menu displays these commands:

- **X&Y Spacing**—Sets the spacing of the grid lines in both the x-direction and y-direction to the same value.
- **X Spacing**—Sets the x-spacing of the grid lines only.
- **Y Spacing**—Sets the y-spacing of the grid lines only.
- **Angle**—Modifies the angle between the screen horizontal and the x-direction grid.

For a Polar grid, the POLAR PARAMS menu displays these commands:

- **Ang Spacing**—Sets the angular spacing between radial lines. The value you type must divide evenly into 360.
- **Num Lines**—Sets the number of radial lines to use. The angular spacing is 360 per number of lines.
- **Rad Spacing**—Modifies the spacing of the circular grid.
- **Angle**—Modifies the angle between the horizontal and the 0-degree radial line.

About the Pro/ENGINEER Drawing Modes

Pro/ENGINEER offers functionality for working with engineering drawings in two separate components: Drawing mode and an optional add on module, Pro/DETAIL.

Using the Pro/ENGINEER Drawing mode, you can create drawings of all Pro/ENGINEER models, or import drawing files from other systems. You can annotate the drawing with notes, manipulate the dimensions, and use layers to manage the display of different items. All views in the drawing are associative: if you change a dimensional value in one view, the system updates other drawing views accordingly. Moreover, Pro/ENGINEER associates drawings with their parent models- the model automatically reflects any dimensional changes that you make to a drawing. In addition, corresponding drawings also reflect any changes that you make to a model (such as the addition or deletion of features and dimensional changes) in Part, Sheetmetal, Assembly, or Manufacturing modes.

Pro/DETAIL

Pro/DETAIL, the optional add-on module, extends the drawing capability offered by Pro/ENGINEER. You can use it with basic Pro/ENGINEER or as a standalone module to create, view, and annotate models and drawings.

Pro/DETAIL supports additional view types and multisheets, offers numerous commands for manipulating items in a drawing, and lets you add and modify different kinds of textual and symbolic information. In addition, you can use it to customize engineering drawings with sketched geometry, create custom drawing formats, and make multiple cosmetic changes to drawings.

With Pro/DETAIL, you can also use a pop-up menu to modify any object in a drawing from anywhere in the menu tree. At any time when a drawing window is active, you can interrupt your current process and activate a drawing object for modification .

Drawing Interfaces

With a license for Pro/INTERFACE or Pro/DETAIL, you can access various interface commands for exporting drawing files to other systems and importing files into Drawing mode.

To Export a Model from Drawing Mode

You can export the following models from Drawing mode to a data file: IGES, DXF, DWG, SET, STEP.

1. Click **File > Save As**. The **Save a Copy** dialog box opens.
2. Under **New Name**, enter the desired new file name.
3. From the **Type** list, select the desired file type to which you want to export the current drawing.
4. Click **OK** to save the drawing and close the **Save a Copy** dialog box.

To Export a Drawing as an Image File

There are several ways to export drawings as image files. You can capture the current screen using the **Save a Copy** command, or you can "plot" any number of the drawing sheets to an image file using the Print command.

You can also use the **File** > "**Save as Picture**" command to create PTC .pic files of sheets at various stages of design, and compare them to current sheets for differences using the **Analysis** > **Compare Sheet to Picture** command.

The following types of image files are supported:

JPG - Compressed file for Web viewing (Plot Only)

TIFF - Detail-rich image file suitable for printing (Screen capture or Plot)

PIC - A proprietary PTC format. used for file previews, picture-to-file comparisons and the Model View program.

To save a TIFF file of the current screen:

1. Click **File > Save a Copy**. The **Save a Copy** dialog box opens.
2. Under **New Name**, enter the desired new file name.
3. From the **Type** list, select TIFF.
4. Click **OK** .You prompted "Pop the current window?" Click **Yes**. The file is created.

To plot a whole sheet (or sheets) as a TIF or JPG file:

1. Click **File** > **Print**. The **Print** dialog box opens.

2. Click **Destination** > **Add Printer Type** > **TIFF** or **JPEG**.

3. Check **To File**.

4. Click OK. The **Print to File** dialog box opens.

5. Use the dialog box to name the new file. Click OK.

6. The new image file is created.

To save a sheet as a .PIC file

1. Click **File > Save as Picture**.

2. The pic file is created, using the drawing name. It is appended with a successive number, for example part.pic.1, part.pic.2, etc.

To Compare a Drawing to a Saved Image File

You can compare a drawing sheet to a previously saved .pic file of the sheet. (Use the **File** > **Save as Picture** command to save .pic files.) You can compare the whole sheet or use any portion of it in magnification.

1. Click **Analysis** > **Compare Sheet to Picture**. The **File Open** dialog box opens

2. Select the .pic file to compare.

   The pic file is "layered" over the drawing page. Common items are suppressed, and any different items, either added or deleted, are shown.

3. To return to the current drawing click Repaint.

About Importing Draft Data from External Applications

You can add draft entities to a Pro/ENGINEER drawing if you have a license for Pro/DETAIL or Pro/INTERFACE. If you have a license for Pro/DETAIL, you can import the following file types:

- IGES
- SET
- DXF
- STEP
- CGM
- STL
- VRML
- DWG
- POLTXT

- ACIS file
- VDA
- Neutral
- CADAM
- CATIA
- PDGS
- ECAD

If you have a license for Pro/INTERFACE, you can import the same files, but you cannot modify the entities if you do not have Pro/DETAIL.

To Import External Draft Entities into the Current Drawing

1. Click **Insert > Data from File**.
2. In the **Open** dialog box, use **Look In** to navigate to the directory that houses the file you want to import, and then select the file.
3. Click **OK**. The system imports the selected file into the current model.

To Create Groups of Entities That Maintain Their Group Associativity

To create a group of entities which maintain their group associativity through an IGES export:

1. On the DRAWING menu, click **Advanced > IGES Groups > Create**.
2. Enter the IGES group name, and then select the items that you want to belong to the group. You cannot select items from different views. When you are done selecting items, click **Done Sel**.

The system creates the group.

About Drawing Setup File Options

Pro/ENGINEER saves drawing setup file options with each individual drawing and drawing format. These setup file options determine such characteristics as the height of dimension and note text, text orientation, geometric tolerance standards, font properties, drafting standards, and arrow lengths.

The system gives default values to these setup file options, but you can modify the values to customize a drawing, and save them to use in other drawings. The system saves (and retrieves) the values in a drawing setup file named `filename.dtl`. The file that you specify in the configuration file option `drawing_setup_file` establishes the default drawing setup file option values for any drawing that you create during a Pro/ENGINEER session. If you do not set this option, the system uses the default drawing setup file option values. If you have a license for Pro/DETAIL, you can install sample drawing setup files for DIN, ISO, and JIS from the loadpoint/text directory with the following names:

- `din.dtl`
- `iso.dtl`
- `jis.dtl`

Retrieve these setup files to set the desired environment in your drawing. You use DRAWING > **Advanced > Draw Setup** on the Menu Manager to create, retrieve, and modify a drawing setup file. This command opens the **Options** dialog box.

Note: A drawing format always has its own setup file—independent of the drawing setup file—but it is restricted.

To Create a Drawing Setup File

Using **Advanced > Draw Setup** on the DRAWING menu, you can create a drawing setup file by saving a copy of the default setup file under a different file name, and then editing the copy. Drawing setup files contain the extension `.dtl`.

1. In the Menu Manager, click DRAWING > **Advanced** > **Draw Setup**. The **Options** dialog box opens and displays the options in the current drawing setup file. The options are organized in the list according to the function that they perform.

2. Save a copy of the current file under a different file name by clicking to open the **Save a Copy** dialog box. Then, save a copy of the current file under a different file name.

3. Click to open the new setup file, and then make edits to the file as desired.

4. After editing the file, click **Apply** to apply your changes, and then click **Close** to exit the **Options** dialog box. The new file is saved to the current working directory and is available for use by other drawings.

5. To update the drawing, do one of the following:
    - Click **View > Repaint**
    - Click the **Redraw** button
    - Click **View > Update > Current Sheet** or **All Sheets**, and then use the GET SELECT menu to select the view you want to update.

    The system updates the drawing using the parameter values of the new drawing setup file.

To Change the Default Text Editor for Drawing Setup Files

To change the default text editor for drawing setup files, you can use the configuration file option `drawing_file_editor`. The default setting is `protab`, which specifies Pro/TABLE as the default text editor.

To use an editor other than the system editor (Pro/TABLE):

1. On the menu bar, click **Utilities > Options** and set the configuration file option `drawing_file_editor` to `editor`.

2. Open the drawing setup file by clicking DRAWING > **Advanced > Draw Set Up** on the Menu Manager. Then, set the drawing setup file option `pro_editor_command`. The value for this option must be the full path to the executable file of the new text editor. For example, if you wanted to specify Notepad as the default text editor, you would type the full path to the executable for the Notepad application in the **Value** box (in the **Options** dialog box). The value might appear as follows:
    `C:\Winnt\notepad.exe`

To Retrieve a Drawing Setup File

You retrieve an existing drawing setup file in the same way you retrieve a configuration file from within the **Options** dialog box. When you retrieve a drawing setup file, Pro/ENGINEER reads in its values to the current (active) drawing.

1. Choose DRAWING > **Advanced > Draw Setup**. The **Options** dialog box opens and displays the options in the current drawing setup file.

2. Click . The **File Open** dialog box opens and displays the files in the current working directory.

3. Choose a drawing setup file from the file list, and then click **Open**. The selected drawing setup file opens and its options display in the **Options** dialog box.

4. Click **View > Repaint**, or click the **Repaint** button, or click **View > Update**. The system updates the drawing using the parameter values of the retrieved file.

Specifying Your Setup Files Directory

Using the configuration file option `pro_dtl_setup_dir`, you can specify the complete path to the directory that contains your drawing setup files. When you choose **Advanced > Draw Setup** from the DRAWING menu, The **Options** dialog box opens and automatically displays the directory that contains your drawing setup files. If you do not specify the pathname using this configuration file option, the system brings you into the default setup directory.

To Modify the Current Drawing Setup File

1. Choose DRAWING > **Advanced** > **Draw Setup**. The **Options** dialog box opens and displays the options for the current drawing setup file.
2. Edit the values as necessary, and then click **Apply** to save your changes.
3. Click **Close** to close the **Options** dialog box.
4. Click **View > Repaint**, or click the **Repaint** button, or click **View > Update**. The system updates the drawing using the new setup file option values.
   Note: If you modify the option values for the current drawing setup file, the system modifies only the current drawing. It does *not* modify drawing setup files on the disk unless you save the current values by choosing **Save**.

To Work in Multiple Windows

You can show multiple drawing sheets in multiple windows (or the same sheet in multiple windows), create a new sheet in a new window, and select any window.

1. Click **Windows > New**.
2. At the prompt, type the sheet number to appear in the new window, and then press ENTER. You can also add a new sheet to the drawing. The new window appears with the specified sheet.

Selecting in Multiple Windows

You can select items in multiple windows using commands in the GET SELECT menu, but keep in mind the following:

- You can select from any active sheet in any window. For example, you can select details to delete in all windows.

- If you select an item in a window other than the current window, the selected window becomes the active window. For example, if you select a dimension in a window, that window becomes active. You can then locate the dimension.

Objects and Procedures that are Changed in All Windows

When you are working with multiple windows, Pro/ENGINEER simultaneously reflects the changes in all windows of the same drawing for the following types of objects:

- Dimensions
- Notes
- Axes
- Symbols
- Datums
- Views

The system also simultaneously reflects the following procedures in all windows of the same drawing:

- Modifying the color of draft entities, cosmetic features, notes, or symbol instances

- Creating or modifying a draft entity
- Manipulating draft datums
- Deleting geometric tolerances

Customizing Your Environment

Many configuration file options and several commands in the **Environment** dialog box control the display of items in a drawing and access to files. **Environment** dialog box settings supersede configuration file settings. Therefore, if you seldom use a particular command (such as the drawing grid), you could use the configuration file option to keep it cleared, and use the **Environment** dialog box button only when you decide to use the drawing grid.

Configuration File Options for Drawing Mode

The following configuration file options are specific to Drawing mode functions. See Configuration File Options in Pro/HELP for full information about these options and for a complete list of configuration file options available for Pro/ENGINEER.

```
allow_move_view_with_move
allow_mod_attach_draft_dim
allow_move_attach_in_dtl_move
auto_associative_dimensions
auto_regen_views
create_drawing_dims_only
default_draw_scale
dim_fraction_format
display_dwg_tol_tags
display_in_adding_view
disp_trimetric_dwg_mode_view
draw_models_read_only
draw_points_in_model_units
drawing_file_editor
drawing_setup_file
drawing_view_origin_csys
dwg_select_across_pick_box
force_wireframe_in_drawings
format_setup_file
highlight_erased_dwg_views
highlight_new_dims
make_parameters_from_fmt_tables
make_proj_view_notes
pro_dtl_setup_dir
pro_format_dir
rename_drawings_with_object
save_drawing_picture_file
save_modified_draw_models_only
selection_of_removed_entities
switch_dims_for_notes
sym_leader_orient_move_text
todays_date_note_format
variant_drawing_item_sizes
```

About Retrieving Drawings in View-Only Mode

If you have a license for Pro/DETAIL, you can significantly reduce the amount of time it takes to retrieve a drawing or report by using *View-Only* mode.

The system does not retrieve any of the associated solid models if you retrieve a drawing by choosing **Open** from the Pro/ENGINEER **File** menu, and then select the **Retrieve Drawing as View Only** check box from the list in the upper-right corner of the **File Open** dialog box. Since the solid models are not in session, the system temporarily freezes the drawings, so you cannot modify them.

View-Only Mode Restrictions

The following restrictions apply to View-Only mode:

- If the geometry display of a view is missing, the system displays an empty view boundary.
- Since Pro/ENGINEER does not retrieve any of the associated solid models, plotting (for example, complex overlap checking) does not function the same way that it does in Drawing mode because much of the information is missing.
- If you retrieve a drawing that is already in memory into the current window in a different mode (for example, the drawing is in View-Only mode, and you retrieve it using **Search/Retrieve**, or vice versa), the drawing remains in its original mode and Pro/ENGINEER displays a warning message.
- You cannot modify or store drawings unless you use **Retr Models** in the VIEWONLY DRW menu to retrieve all of the models.
- If you store the display with snap lines in Drawing mode, the system plots them in View-Only mode.
- On a multisheet drawing, if you retrieve only one sheet, but you want to store the display, the system gives you only the display of one sheet because it has not yet regenerated the other views.

  To regenerate all sheets, choose **View > Update > All Sheets** on the Pro/ENGINEER menu bar. You must do this manually before storing the drawing if you want the View-Only functionality to work for all sheets.
- If you save the display (by selecting **Save Display** from the **Environment** dialog box or setting the configuration file option `save_display` to `yes`), Pro/ENGINEER does not have to calculate the view display. As a result, you can significantly reduce the retrieval time in Drawing mode.

Tip: Saving the Display

If you set the configuration file option `save_display` to `yes`, the system stores view geometry and detail items such as solid dimensions, and displays them when it retrieves the drawing in View-Only mode. If you set this option to `no`, it does not display them. However, instead of modifying your configuration file, you can select **Save Display** from the **Environment** dialog box since it performs the same function.

To Display a Drawing in View-Only Mode

1. Save existing display information by selecting **Save Display** from the **Environment** dialog box or setting the configuration file option `save_display` to `yes`. To regenerate an entire view or the display for the current sheet (or all sheets), click **View** > **Update**.
2. Click **File** > **Save** to save your drawing with the display information you need.
3. Click **File** > **Open**; then click and select the **Retrieve Drawing as View Only** option from the list.
4. Retrieve the drawing. The system displays the drawing in View-Only mode. Choose **Sheet** from the VIEWONLY DRW menu to switch to another sheet.
5. To determine if any display information is missing, click **Info > Session Info**. If all information is there, the system displays a message. If any is missing, it displays a text window, identifying the missing information.

To Modify a Drawing in View-Only Mode

1. Choose **Retr Models** from the VIEWONLY DRW menu
2. Click **Confirm**. The system enters Drawing mode and retrieves all models for the current drawing.
3. Modify the current drawing as necessary.

About Storing Drawings

Pro/ENGINEER saves some of the entities and information that you can create or modify in a drawing with the model, rather than with the drawing. This is important, since such changes made to the drawing could affect the model that it documents. For example, when you set a dimension as basic, the dimension becomes theoretically exact, and the system removes any of its tolerances.

Pro/ENGINEER saves with the *drawing* strictly cosmetic information such as the following:

- All draft entities
- The view in which an entity appears
- The placement of an entity on the sheet
- Jogs and breaks in leaders and dimension lines
- The insertion of dimensions in notes
- The font, height, width, and slant angle of text

However, it saves with the *model* much of the information you add to a drawing, such as the following:

- Geometric tolerances (you can also save them in the drawing)
- Dimension information (also reference and driven dimensions), including the following:
  - Additional text
  - Standard/ordinate dimension type
  - Attached geometric tolerance list
  - Attached set datum or axis reference
  - Value and tolerance information
  - The difference between the primary and secondary units, when you have explicitly set them
  - Basic and inspection attributes
  - Set datum and axis information
  - Datum target point information
  - Surface finishes, including type and value
  - Simplifications (using **By View** and **All Views**)
  - Layer membership information for all entities in the model

To Disallow Changes Affecting the Model

To disallow changes made in Drawing mode that affect the model, set the configuration file option `draw_models_read_only` to `yes`. In this case, when you attempt to make a change that affects the model, Pro/ENGINEER issues a warning and does not make the modification.

To Save the Drawing without Storing the Model

Whenever you save a drawing after making changes that affect the model, Pro/ENGINEER saves the model with the drawing. However, if you do not make changes to a model, and you want to save the drawing without storing the model, set the configuration file option `save_modified_draw_models_only` to `yes`.

About Regenerating Views and Drawings

When you regenerate a drawing, Pro/ENGINEER recreates the drawing and the model that it represents; it does not simply redraw them. Consequently, if you changed any dimension values in the model within Drawing mode, the system updates the model to reflect these changes when you regenerate the drawing. The regenerated drawing displays the updated model and any changes that you made to it.

If you create driven dimensions or draft dimensions, you must regenerate the drawing after you move draft entities and when you want to show changes made to the drawing setup file option `draft_scale`.

Regenerating the drawing updates the driven dimensions and snaps back the associated dimensions to the draft entities. If you use driven dimensions in model relations, you must regenerate the model in the drawing to update them.

Regenerating a View

The **Update** command in the Pro/ENGINEER **View** menu works together with the configuration file option `auto_regen_views`. The options for **Update** are as follows:

- **Drawing View**—Allows you to select the view(s) you want to regenerate.
- **Current Sheet**—Regenerates the current sheet.
- **All Sheets**—Regenerates all sheets in the drawing.

The auto_regen_views Configuration File Option

When you set the `auto_regen_views` configuration option to `yes`, the system automatically updates the drawing display by a repaint when you change from one window to another, such as when you modify a model in a subwindow while you are working on a drawing in the main window. You can repaint or regenerate the drawing to reflect changes made to the model. When you regenerate it, the system updates the model to reflect the changes made in the drawing.

If you set `auto_regen_views` to `no`, you can update only the drawing by choosing **Update** from the **View** menu, and then selecting **Drawing Views**, **Current Sheet**, or **All Sheets**. Neither the **Update** command in the **View** menu nor the **Regenerate** command in the DRAWING menu updates the drawing when you have this option set to `no`, even if you make the change to the model in Drawing mode (such as modifying a dimension value). You can select as many views as you want to regenerate at the same time.

If you try to modify a view that you have not updated, Pro/ENGINEER displays an error message informing you that it is not going to make changes to the drawing until you apply the **Update** or **Regenerate** command to the view.

Note: When you regenerate a parent view, its child views do not automatically regenerate; you must individually select each view on the drawing, including detail views. Whenever you save changes to the model, the system displays them on the drawing the next time that you retrieve it, regardless of whether you regenerated the drawing views.

To Update a Drawing View

To update (regenerate) a drawing view after making changes to its source model,

- Click **View** > **Update**, and then choose one of the following options:
  - **Drawing Views**—Regenerates the selected views. Select the views you want to regenerate, then click **Done Sel** from the GET SELECT menu.
  - **Current Sheet**—Regenerates all views on the current sheet only.
  - **All Sheets**—Renerates all views on all sheets in the current drawing.

Note: Regenerating a drawing view is not object-action oriented. That is, you cannot regenerate a view by first selecting the view and then choosing **View > Update**. You must choose the command first, and then select the view(s) you want to regenerate.

To Regenerate a Model or Draft Dimensions

Whenever you change a model, use the **Regenerate** command on the DRAWING menu to regenerate it and to update drawing views accordingly. Using this command, you can also update associative draft dimensions when draft entities change.

1. Choose DRAWING > **Regenerate**.
2. Do one of the following:

- To update the model and redraw all views accordingly, choose REGENERATE > **Model**.
- To regenerate only associative draft dimensions, choose REGENERATE > **Draft** (this command does *not* regenerate a model, even if you change it).

About Drawing and View Scales

You can use two types of scales in a drawing: global, or drawing scale (when adding views using the **No Scale** command) and individual scale (when adding views using the **Scale** command).

- With a *global* scale (or drawing scale), the system scales drawings according to the drawing SCALE parameter at the bottom of the drawing sheet. For example, a value of .25 means that it scales the drawing views at one-quarter of the actual size of the model. You can set a default drawing scale using the configuration file option `default_draw_scale`, or you can override your settings by modifying the scale.
- With an *individual view* scale, the system gives views an individual scale factor that appears below each view, shown in a note "Scale *value*." When you modify the drawing scale, these views do not change, since the scale factors are independent.

When you change the size of the drawing, the drawing scale changes to match, to keep the views in proportion to the size of the sheet. However, detailed and scaled views retain their original scale regardless of changes to the drawing size.

View Scales Driven by Relations

You can use relational expressions to drive a view scale of a view that you placed using the **Scale** command; however, you cannot use them to drive any views that you scale globally using the **No Scale** command. When you type a relational expression for the view scale, the system calculates the expression and retains the information. This functionality enables you to relate the view scale using dimensions within the model. The system stores the relation with the model and updates the view scale each time the dimension in the model changes, along with all items related to it.

Note: When you rescale a scaled drawing view, Pro/ENGINEER also rescales all related parent/child views (projected views, broken views, etc.).

To Modify the Relational Expression for the View Scale

1. On the menu bar, click **Edit > Value**.
2. Select the view scale in the lower-left corner of the screen.
3. Type the real value or relational expression and press ENTER.

Note: When using a relational expression, type *only* the right-hand side of the expression. The system calculates the left-hand side as the view scale.

Drawing Scale Format

You can express drawing scales in a decimal or fractional format by setting the drawing setup file option `view_scale_format` (`decimal format` is the default). To establish the pattern of fractional scales, use the drawing setup file option `view_scale_denominator`.

About Multisheet Drawings

If you have a license for Pro/DETAIL, you can use the **Sheets** command in the DRAWING menu to create multiple sheet drawings and move items from one sheet to another. You can view the sheets of a multisheet drawing using basic Pro/ENGINEER. If a drawing has more than one sheet, an additional tag, SHEET # OF *[]*, appears at the bottom of the main window.

When working with multisheet drawings, keep in mind the following:

- When you switch a projection view to another sheet, it becomes independent. You can then move the parent view, and the system does not update the projection view on another sheet. If you switch these views again to the same sheet, the projection view immediately becomes a child of the parent.
- You can change drawing scales on each sheet independently.
- When you change the scale on a sheet, partial views on other sheets do not change.
- If you erase a view on one sheet, you can resume it on any other sheet.
- You can change which sheet is currently active by using the Sheet spin box located on the toolbar.

To Add a New Sheet to a Drawing

On the DRAWING menu, click **Sheets > Add**. Pro/ENGINEER automatically adds a new sheet to the end of the drawing.

To Remove Sheets from a Drawing

1. On the DRAWING menu, click **Sheets > Remove**.
2. Enter the sheet numbers of the sheets you want to remove. You can separate them by using commas or dashes and insert spaces anywhere, for example, `[1,3-5,12,15-20]`. The number to the left of a dash must always be less than the number to the right. Press ENTER when you are done.
   The system removes the specified sheets from the drawing. If it removes the current sheet, it sets the first remaining sheet as the current sheet.

Tip: If Sheet Removal Automatically Cancels

Pro/ENGINEER cancels the procedure and does not remove any sheets if you perform any of the following actions:

- Press ENTER without specifying any numbers.
- Use incorrect syntax in the input line.
- Type pairs of numbers without putting them in ascending order and separating them by a dash.
- Include nonexistent sheets (however, you can include the same sheet more than once; for example, `[1-5,3-8]` is the same as `[1-8]`).
- Specify the removal of all of the sheets of a drawing (if only one sheet is present, the **Remove** command is not available).
- Specify the removal of a sheet that holds the parent view of a child view shown on another sheet that is kept (however, you can remove both sheets at once).

To Reorder the Sheets in a Drawing

1. On the DRAWING menu, click **Sheets > Reorder**. Pro/ENGINEER displays the range of page numbers for the drawing.
2. Type a new number for the current sheet. The system rearranges the sheets of the drawing, inserting the current sheet into the specified position in the drawing.

To Move Items to Another Sheet

Using the **Switch Sheet** command in the SHEETS menu, you can move items to another sheet.

1. On the DRAWING menu, click **Sheet > Switch Sheet**. The DWG ITEMS menu opens and displays the following commands:
   - **Dwg Views**—Switches drawing views.
   - **Draft Items**—Switches detail items without leaders.
   - **Dwg Tables**—Switches drawing tables.
2. Choose a command; then select items to switch.

3. To confirm your selection, choose **Done Sel**. The system highlights the selected items in magenta.
4. Choose DWG ITEMS > **Done** and type the destination sheet number. You can also start a new sheet on the fly by entering a new sheet number. If you type the current sheet number for the destination sheet, the system prompts you to retype the number. Press ENTER when you are done.
5. The items appear on the specified sheet. To continue switching items, choose **Sheet > Switch Sheet** and repeat the process.

Tip: Moving Draft Items

You cannot move draft items that have leaders attached to entities such as balloons, notes, and dimensions. If the drawing has only one sheet, the system automatically creates a second sheet after you choose items to move.

To Move Items to Any Place on Another Sheet

1. On the DRAWING menu, click **Sheets** > **Switch Sheet >Switch/Move**.
2. Choose a command from the DWG ITEMS menu.
3. Select items to switch. To confirm your selection, choose **Done Sel**. The system highlights the selected items in magenta.
4. Choose DWG ITEMS > **Done** and type the destination sheet number. Press ENTER.
5. Select a method of defining the translation vector by choosing the appropriate commands from the GET VECTOR menu:
6. Items appear on the specified sheet at the indicated location. To continue switching items, choose **Sheets > Switch Sheet** and repeat the process.

The GET VECTOR Menu

The GET VECTOR menu allows you to translate (move) items to any place on another drawing sheet by defining a translation vector. For information on moving items to another sheet, refer to To Move Items to Any Place on Another Sheet in Pro/HELP.

The following options for defining a translation vector are available on the GET VECTOR menu:

- **Horiz**—Translates the entities along the horizontal direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the right of the sheet.
- **Vert**—Translates the entities along the vertical direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the top of the drawing sheet.
- **Ang/Length**—Translates the entities at an angle, and a specified distance in that direction, measuring the angle relative to the horizontal in the counterclockwise direction. An arrow appears showing the positive direction of translation.
- **From-To**—Translates the entities along a vector defined by using a start point and endpoint.

Maintaining Size and Position of Drawing Models

When moving items from one sheet to another in a two-dimensional drawing model, use the configuration file option `variant_drawing_item_sizes` to control whether drawing items maintain their size and position on paper if you change the sheet size or drawing units.

- If you set this option to `yes`, some items scale and/or reposition to be the same size or in the same place when you plot them on paper, while other items scale and/or position themselves to be the same size or in the same place on screen.
- If you set it to `no`, items that you move or copy to a different sheet of the same or different 2-D model (or items on a sheet that changes size and/or units) retain the same size on paper and the same relative orientation as they did previously.

About Drawing Templates

Drawing templates are used when creating a drawing. They automatically create the views, set the desired view display, create snap lines, and show model dimensions based on the template.

Drawing templates contain three basic types of information for creating new drawings. The first type is basic information that makes up a drawing but is not dependent on the drawing model, such as notes, symbols, and so forth. This information is copied from the template into the new drawing.

The second type is instructions used to configure drawing views and the actions that are performed on that view. The instructions are used to build a new drawing with a new drawing object (model).

The third type is parametric notes. Parametric notes are notes that update to new drawing model parameters and dimension values. The notes are re-parsed or updated when the template is instantiated.

Use the templates to:

- Define the layout of views
- Set view display
- Place notes
- Place symbols
- Define tables
- Create snap lines
- Show dimensions

You can also create customized drawing templates for the different types of drawings that you create. For example, you can create a template for a machined part versus a cast part. The machined part template could define the views that are typically placed for a drawing of a machined part, set the view display of each view (for example, show hidden lines), place company standard machining notes, and automatically create snap lines for placing dimensions. Creating a template allows you to create portions of drawings automatically, using the customizable template.

To Create a Drawing Template

1. Click **File > New**. The **New** dialog box opens.
2. Click **Drawing**, and then type the name of the template you are creating or accept the default.
3. Select the **Use default template** checkbox (selected by default), and then click **OK**. The **New Drawing** dialog box opens.
4. Click **Empty** or **Empty with format**.
   If you click **Empty with format:** Under **Format**, specify the format you want to use. Then click **OK**. Pro/ENGINEER creates a new drawing with the specified format.
   If you click **Empty:** Under **Orientation**, specify the template orientation by clicking **Portrait**, **Landscape**, or **Variable**. For **Portrait** or **Landscape**, choose a standard size under **Size**. For **Variable**, specify a size using the **Width** and **Height** boxes, and specify a unit by selecting **Inches** or **Millimeters**.
5. Click **OK** to create the template.
6. Click **Applications > Template** to enter Drawing template mode, and then in the TMPLT DWG menu, click **Views > Add Template**. The **Template View Instructions** dialog box opens.
7. Type the View Name or accept the default, and then specify the view orientation.
8. In the **Model "Saved View" Name** text box, orient the view.
9. Specify view options and view values in the **View Options** and **View Values** areas.
10. Click **Place View** and select the location of the General view.
    Note: After you place the view, you now have the options to move the symbol, edit the view symbol, or to replace the view symbol.
11. To place additional views, click **New**, type the new view name, and orient the new view. Specify the view options and view values of the new view.
12. When you are done placing all of the desired views, click **OK.** Save the template.

To Create a Drawing Using a Drawing Template

1. Click **File > New**. The **New** dialog box opens.
2. Click **Drawing**, and then type the file name for the drawing you are creating or accept the default name.
3. Clear the **Use default template** checkbox, and then click **OK**. The **New Drawing** dialog box opens.
4. Select the model from which you want to create the drawing.
5. Specify the template by clicking **Use template**. Type the name of the template you want to use or select a template from the Template list. Click **OK**. The drawing is created.

Note: The views with the correct attributes in both the template and the model are created. If attributes that are defined in the template do not exist in the model, errors occur when the drawing is being created. The **Drawing Template Error Info** dialog box opens and lists the errors.

To access the **Drawing Template Error Info** dialog box, click **Info > Template Errors**.

The Template View Instructions Dialog Box

The **Template View Instructions** dialog box is accessed from the TMPLT DWG menu (on the Menu Manager), which appears when you click **Application > Template** in Drawing mode. For more information about Drawing template mode, refer to To Create a Drawing Template in Pro/HELP.

Use the following options in the **Template View Instructions** dialog box to customize your drawing templates:

- **View Name**—Set the name of the drawing view that will be used as the view symbol label.
- **View Orientation**—Create a General view or a Projection view.
- **Model "Saved View" Name**—Orient the view based on a named view in the model.

View Options	View Values
X-Section—Set cross-section. Only full, total cross-sections are supported	X-Section Name Arrow Placement View
Scale—Enter the scale value or no scale.	View Scale
Explode State—Set the exploded view.	Explode Name
Simplified Rep—Set the view as a simplified representation. If it is not a simplified representation, the default is a master representation.	Simplified Rep Name
Process Step—Set the process step for the view.	Step Number
Model Display—Set the view display for the drawing view.	Wireframe, Hidden Line, No Hidden, Default
Tan Edge Display—Set the tangent edge display.	Tan Solid, No Disp Tan, Tan Ctrln, Tan Phantom, Tan Dimmed, Tan Default
Snap Lines—Set the number, spacing and offset of the snap lines.	Number, Incremental Spacing, Initial Offset
Dimensions—Show dimensions on the view.	Create Snap Lines, Incremental Spacing, Initial Offset

Balloons—Show balloons on the view.
- **Place View**—Places the view after you have set the appropriate options and values.
- **Edit View Symbol**—Allows you to edit the view symbol using the **Symbol Instance** dialog box.
- **Replace View Symbol**—Allows you to replace the view symbol using the **Symbol Instance** dialog box.

About Assembly Drawings

In an assembly drawing, you can show and erase assembly and part dimensions. You can also create dimensions on individual components directly in an assembly view. However, keep in mind the following:

- You can display parameters for assembly features and all assembly components in assembly drawing notes, but you *cannot* type part parameters.
- The dimensions in an assembly drawing are visible for modification *only* if the assembly for which the drawing was created is in session.

Resuming Cosmetics

When you suppress parts or subassemblies in an assembly drawing, the system retains information about certain cosmetics of these views. When you resume them, you also resume these cosmetics:

- Dimension placement
- Surface finish symbol placement
- Parametric text in notes and symbol instances
- Cross-hatching cosmetics
- References to cross-sectional edges
- Detail axis cosmetics
- Member, edge, and set datum display information
- Draft geometric tolerances that reference part datums

About Getting Drawing Information

By using the **Info**, **Edit**, and **Analysis** menus, you can access information about a drawing. You can:

- Highlight currently displayed items using filters. By using **Edit > Find in Sheet**, you can filter the display of specified items according to item type (such as dimension, note, or geom tol), layer, and view.
- Perform measurement analyses on draft entities that exist in the drawing, by using **Analysis > Measure Draft Entities**. You can get information about distance, angle, intersection point, and tangent point.
- Display geometric and cosmetic information about an entity, and save the information to a file, by using **Info > Draft Entity**.
- Save a note as a text file, by using **Info > Save Note**.
- Display the grid angle and spacing values, by using **Info > Draft Grid**.
- Obtain information about out-of-date displays in the drawing, by using **View > Check Display Status**. You can save the information to a file and select a recommended action to update the drawing (such as repainting and regenerating).
- Obtain information about why a template failed when you attempted to open it, by using **Info > Template Errors**.

To Get Information About Draft Entities

Use the following procedure to obtain cosmetic and geometric information about a selected draft entity, and to save the information on the screen or to a file.

1. Click **Info > Draft Entity**.
2. Select a draft entity using the GET SELECT menu.
   An Information window appears containing information about the selected draft entity.
3. To save the information on screen or to a file, click **File > Save As**.

To Perform Measurement Analyses on Draft Entities

1. Click **Analysis > Measure Draft Entities**. The **Draft Measure** dialog box opens, and the GET SELECT

menu appears.

2. In the **Type** list, select the type of draft to measure (**Distance, Angle, Intersection Point,** or **Tangent Point**).
3. Under **From** and **To**, select the first and second entities to measure.

   Note: To redisplay the GET SELECT menu, click the **Select** button [image].
4. If you selected **Angle**, you can select **Use horizontal** to calculate the angle, instead of selecting a second entity.
5. Click **Compute**. The computations appear under **Results**. The results include `Angle`, `Distance`, `Intersection Point`, and `Tangent point`.
6. Click **Compute** to perform another analysis, and **Info** to display the results in an Information window.
7. Click **Close** to remove the **Draft Measure** dialog box.

To Save a Drawing Note as a File

Use the following procedure to save drawing notes as .txt files.

1. Click **Info > Save Note**.
2. Select the note you want to save.
3. At the prompt, type the file name for the note, and then press ENTER. You do not need to type the file extension; the system inserts it automatically.
4. The note is saved in your working directory.
5. Click **Quit Sel** to exit the GET SELECT menu.

To Display Drawing Grid Information

Use this procedure to display the grid angle and spacing values for a grid in a drawing.

• Click **Info > Draft Grid**.

The angle and spacing values for the grid are displayed in the message area.

To Get Information About Out-of-Date Displays in a Drawing

1. Click **View > Check Display Status**. An Information window appears displaying the sheet number, view ID, view name, any information about missing display status, and the recommended action to update the drawing.
2. To save the information to a file, click **File > Save As** in the Information window.

To Get Information About Drawing Template Failures

Use this procedure to obtain information about why a drawing template failed when you attempted to open it.

• Click **Info > Template Errors**.

An Information window opens displaying the errors encountered when you attempted to open the template.

To Highlight Items by Type and Attributes on the Current Sheet

Use this procedure to highlight displayed items on the current sheet, using the type and attributes filters provided in the **Highlight by Attributes** dialog box.

1. Click **Edit > Find in Sheet**. The **Highlight by Attributes** dialog box opens.
2. Under **Item Type**, select the types of items you want to highlight.

3. Under **Owner Model**:
   - To highlight items that were created in the drawing (but not in the associated model or models), select **The drawing**.
   - To highlight items that were created in the current model, and shown in the drawing, select **A 3D model**. In a multimodel drawing, the system highlights only items that were created for the current model.

     Both options are selected by default.
4. Under **On Layer**, select one of the following options:
   - **Any or none**—Highlights the specified items, whether or not they exist on a layer.
   - **At least one**—Highlights only specified items that exist on at least one layer. Items that are not on any layer are not highlighted.

     Selected and **Select Layers**—Highlights only specified items that are on the layers you specify.
5. Under **Dimension Type**, select the attributes of the type of dimensions you want to highlight.
   - **Owned by a model**—Highlights dimensions that were created and reside in the associated model or models (such as assembly and feature dimensions). In multimodel drawings, all dimensions that reside in all associated models are highlighted.
   - **Created and associative**—Highlights dimensions that were created in the drawing and are associative; that is, they update when the referenced entity is regenerated (resized).

     Note: If you have set the drawing setup file option `associative_dimensioning` to `no`, this option is disabled.
   - **Created but non-associative**—Highlights dimensions that were created in the drawing, but are non-associative; that is, they do not update when the referenced entity is regenerated.
6. Under **Displayed in View**, select **Any** to display all items of the specified type in any and all views. Select **Selected** or click **Select View** to select a view in which to display the items, using the GET SELECT menu.
7. Click **Highlight** to highlight the specified items. The items are highlighted in magenta. Click **Close** to close the dialog box.

About Standard Formats

You can use a standard or a sketched drawing format in any number of drawings, and then modify it or replace it at any time. A format is a customized layout for a drawing sheet. It includes a title block, a border, tables, and your company logo. To create standard formats, you must be in Format mode and you must have a license for Pro/DETAIL. You can select the desired format size from a list of standard drawing sizes, or create a new size by specifying values for length and width. To create sketched formats, you must use Sketcher mode in basic Pro/ENGINEER. Since you can modify them parametrically, you can create nonstandard size formats or families of formats.

Standard formats consist of note text, symbols, tables, and draft geometry, including draft cross sections and filled areas. You create or modify a standard format as you would a drawing, and you also use drawing setup files. If you have a Pro/DETAIL license, you can do the following in Format mode:

- Create draft geometry and notes.
- Move, mirror, copy, group, translate, and intersect geometry.
- Use and modify the draft grid.
- Specify user attributes.
- Create drawing tables.
- Use interface tools to create plot, DXF, DWG, SET, TIFF, CGM, and IGES files.
- Import IGES, DXF, and SET files into the format.
- Create user-defined line styles.
- Create, use, and modify symbols.
- Include draft cross sections in a format.

To Create a Standard Format

1. From the Pro/ENGINEER menu bar, choose **File** > **New**.
2. In the **New** dialog box, click **Format** and type a format name in the **Name** box, or accept the default name. Click **OK**.
3. In the **New Format** dialog box, specify the format size by doing one of the following:
   - Under **Orientation**, click **Portrait**, **Landscape**, or **Variable**.
   - If you selected **Portrait** or **Landscape**, select a size from the **Standard Size** list.
   - If you selected **Variable**, you must define both the height and width dimensions. Select **Inches** or **Millimeters** and type values in the **Width** and **Height** boxes.
4. Click **OK**. The system displays the format as specified, and the FORMAT menu appears.

To Create Format Geometry

To create format geometry, you use draft entities. To sketch draft geometry, use the **Sketch** menu, or use any of the sketcher icons in the side menu.

Sheet Outline

The sheet outline is the border of the standard drawing format you selected, as shown in the following figure. Because it is the actual border, it may not appear on pen plots unless you use a paper size larger than the drawing size. Everything within the sheet outline border is also plotted, but you should make an allowance for the plotter hold-down rollers.

To Modify a Standard Format

To modify the values of symbol text, symbol height, and note text in a standard drawing format, use the same commands that are available during format creation.

1. Click **File > Open**, and retrieve the format using the **File Open** dialog box.
2. Use the options on the **Format** menu in the menu bar (Text Style, Arrow Style, Line Style, and so forth).

Placing Parametric Notes in a Format

When you place a parametric note in a format, the note in the format acquires the appropriate value when you use it in a drawing. For example, if you create the note &model_name in a format, the system displays the actual model name in the note.

For the system to update these parameters when you add the format to a drawing, you can include in parametric notes only the drawing labels listed in Appendix B, System Parameters for Drawings (except &todays_date). You must include all user-defined model and drawing parameters in format table cells in the form of &param using **Enter Text** in order for the system to update them in a drawing. Format mode interprets the following types of parametric notes:

- Notes with symbol instances
- Notes with standard system symbols
- Notes with drawing labels
- Notes with a default tolerance

To Add a Table to a Standard Format

Using the **Table** command in the FORMAT menu, you can include tables in drawing formats. When you add a format to a drawing, Pro/ENGINEER copies all of the tables in the format into the drawing. After it copies them, the tables are independent of the original format, and you can move, modify, or delete them.

1. Make sure that the format file to which you want to add a table is the active file.

2. On the FORMAT menu, click **Table > Create**. The TABLE CREATE menu opens.
3. Choose from the following options on the TABLE CREATE menu:
   - **Descending** or **Ascending**—Lets you choose the direction in which to create rows from the starting point.
   - **Rightward** or **Leftward**—Lets you choose the direction in which to add columns from the starting point.
   - **By Num Chars** or **By Length**—Lets you define the length of columns and rows using drawing units (such as inches or millimeters) or by specifying the number of characters as the unit.
4. Use the options on the GET POINT menu to locate the starting corner of the table on the format. The starting corner depends on the choices you made in the TABLE CREATE menu, as specified in the preceding step.
   - **Pick Pnt**—Lets you select a point on the format and specify it as the starting corner.
   - **Vertex**—Lets you select a vertex and specify it as the starting corner.
   - **On Entity**—Lets you select an entity on the format and specify it as the starting corner.
   - **Rel Coords**—Lets you specify relative coordinate values for placing the starting corner by entering relative values for the X- and Y-Axes.
   - **Abs Coords**—Lets you specify absolute coordinate values for placing the starting corner by entering absolute values for the X- and Y-Axes.
5. At the prompt, enter the width of the first column in drawing units, and then press ENTER.
6. Continue entering the width of additional columns until you have the number of columns you want. Then press ENTER again without typing another column width value.
7. At the prompt, enter the height of the first row in drawing units, and then press ENTER.
8. Continue entering values for the height of additional rows until you have the number of rows you want. Then press ENTER again without typing another row height value.

The table displays on the format sheet.

About Using Tables in a Standard Format

Using the **Table** command in the FORMAT menu, you can create tables and add them to drawing formats. Then, when you add a format to a drawing, Pro/ENGINEER copies all of the tables in the format into the drawing. The copied tables are independent of the original format, and you can move, modify, or delete them.

When you **Add/Replace** a format or **Remove** it (using **DRAWING > Sheets > Format > Add/Replace** (or **Remove**), the system highlights the table copied from the old format and asks if it should remove the table. You can choose one of the following: (**Y**)es, (**N**)o, (**K**)eep all, (**R**)emove all.

If you add a table to a format, the drawings that already reference this format do not display the new table. The table must be part of the format when you initially copy it into the drawings. To use the modified format in the drawing, you must choose **Sheets** from the DRAWING menu, **Format** from the SHEETS menu, and **Add/Replace**.

When you add a format to a drawing that contains the format table, the system stores the values you specified for the table as drawing parameters if you have set the configuration file option `make_parameters_from_fmt_tables` to `yes`. You can access the parameters using the **Parameters** command in the ADV DWG OPTS menu. If you set this configuration option to `no`, when you add a format to a drawing, the system prompts you to retype all of the values every time, and it does not evaluate the values on subsequent sheets of the drawing in the format table. These values are nonparametric text. For the system to reprompt you for the parameter text, you must choose **Sheets** from the DRAWING menu, **Format** from the SHEETS menu, and **Add/Replace**.

Note: You cannot use reserved model parameters. If you set `make_parameters_from_fmt_tables` to `yes` in the configuration file, Pro/ENGINEER does not prompt you to type a value for reserved model parameters because it cannot add them to the drawing.

If you change the format size of a drawing with a table, the system scales the table (in the same way that it scales any views and draft entities) to maintain its location relative to other items and the sheet boundaries. Since the drawing setup file option `drawing_text_height` controls the height of any text in the table,

Pro/ENGINEER does not scale text that you include in the table. Therefore, it does not maintain a size that is proportional to the rest of the table. You must modify the text height manually.

To Use Parameters as Labels in a Format Table

You can include parametric labels, such as the drawing name, model name, and sheet number, as text in a format table. For a multimodel drawing, you can type into the format table any parameters related to both the first model and the second model. To include the parametric labels in the table:

1. On the FORMAT menu, click **Table > Enter Text**.
2. Select the table cell where you want to place the label text.
3. At the prompt, enter the appropriate label (such as &dwg_name) and press ENTER. To create additional lines of text, continue entering text for each line. When you are done entering text, click the middle mouse button. The label appears in the table exactly as you typed it until you add the format to a drawing.

Guidelines for Using Parametric Labels in a Format Table

The following rules apply to including parameters as labels in format tables:

- Pro/ENGINEER correctly evaluates parametric labels that you include in a format table *only* if you create the drawing first, add the model, and then add the format. It does *not* evaluate the labels correctly if you create the drawing, add the format, and then add the model.
- When you add a format to a drawing with more than one model present, parametric notes can reference *only* the active drawing model.
- To include text in a table as a title block, use the **Enter Text** command in the FORMAT > TABLE menu. If you move the table, the system does not keep the text that you add as a note with the table.

When you add the format to a drawing, Pro/ENGINEER parses any and all that are present of the standard parametric symbols that it supports and displays the correct values in the table. It does this for every sheet of the drawing on which you use the format, so that the drawing name, model name, or any other standard parameter that you used appears on every sheet that uses that format.

To Give a Format the Same Parameter Values as the Drawing

Each drawing format that you create has its own setup file—one that is completely independent of the drawing setup file. The configuration file option format_setup_file assigns a specified setup file to each format that you create, but does not do so for drawings. For these two setup files to have the same values, you must edit them separately. To give a format the same parameter values as a drawing to which you are adding it:

- Retrieve the drawing setup file into the format using the **Set Up** command in the FORMAT menu. The system reads only those format setup file options that formats are using.

Format Setup File Restrictions

The format setup file is restricted. You can use only the following options:

- drawing_units (you *cannot* modify this option)
- drawing_text_height
- draw_arrow_style
- draw_arrow_length
- draw_dot_diameter
- draw_attach_sym_width
- draw_attach_sym_height

- `leader_elbow_length`
- `default_font`
- `aux_font`
- `text_width_factor`
- `line_style_standard`
- `node_radius`
- `yes_no_parameter_display`
- `sym_flip_rotated_text`

To Reuse a Format from a Legacy System

To reuse an existing format (that is, one that was created in another system), use the interface options (such as DXF, SET, IGES, TIFF, and so on) to import it into your format.

To Save a Standard Format

Before you can use standard drawing formats in a drawing, you must save them. To save a format:

- Choose **Save** from the Pro/ENGINEER menu **File** menu; then type the format name and press ENTER.

About Sketched Formats

To create a sketched format, you must be in Sketcher mode. When you create it, Pro/ENGINEER gives it the name `filename.sec`. The first time you add the format to a drawing, it makes a copy of it in memory and gives it the file extension `.frm`.

When Pro/ENGINEER creates `filename.frm`, that file ceases to have any relationship or associativity with the section from which it was created; therefore, `filename.frm` does not reflect changes that you make to `filename.sec`, and vice versa.

When you add a sketched format to a drawing, the system aligns the lower-left corner (the origin) of the format to the lower-left corner (the origin) of the drawing, and then centers all items in the drawing on the new sheet in the locations that correspond to their positions on the original sheet. If necessary, it adjusts the drawing scale, maintaining relative distances between items.

Note: To copy a file, you must save the `filename.frm` file by choosing **Save As** from the Pro/ENGINEER **File** menu, and typing the format name.

To Create a Sketched Format

1. From the Pro/ENGINEER menu bar, choose **File** > **New**. The **New** dialog box opens.
2. Select **Sketch**. In the **Name** box, type the name of the sketch to use for the format or accept the default file name. Click **OK** to close the **New** dialog box and create the sketch.
3. Sketch the boundary, title block, and so on, and then dimension them.
4. When Pro/ENGINEER successfully regenerates the section, save it using **File** menu commands. You *cannot* save text with a sketched format.
5. Create a drawing and add the sketched format to the new drawing by doing one of the following:
   - During creation of the drawing, in the **New Drawing** dialog box, select **Empty with format**, and then click **Browse** to find and open the sketch name. Click **OK** to create the drawing with the sketched format.
   - After creating the drawing, choose DRAWING > **Sheets** > **Format** > **Add/Replace**. Choose the sketch name (set the `type` filter to `sketcher`).

Effects of Format Size

Pro/ENGINEER automatically assigns a correct paper size to a sketched format. For example, if a format is 7x10 inches, it automatically places the format on an A-size sheet. It does this even if the drawing was originally created in another size. If you add a format of a different size to the drawing, the drawing assumes the size of the format. It scales all views and draft entities accordingly.

Valid Format Extensions

Using Sketcher mode, you can retrieve and change a sketched format. When replacing formats, Pro/ENGINEER looks first for a format with a `.frm` extension, and does not accept `.sec` as a valid extension for format names.

To Replace an Existing Format with a Modified Format

To replace the existing version of the format with the modified one, do one of the following:

- Rename the sketch of the modified format, so that the name is different from the current format, and then use the **Add/Replace** command (DRAWING > **Sheets** > **Format** > **Add/Replace**) to replace the modified sketch in the drawing.
- Use the **Remove** command (DRAWING > **Sheets** > **Format** > **Remove**) to remove the existing format and rename its `.frm` file. Add the modified sketch to the drawing.

Note: You *must* rename one of the formats; otherwise, if you try to replace the current format, it simply replaces itself with the same `filename.frm`.

To Modify and Replace a Sketched Format in Sketcher Mode

1. Retrieve the sketch of the format into Sketcher mode (using **File > Open**). Pro/ENGINEER cannot retrieve a `filename.frm` into Sketcher mode, so `filename.sec` must exist.
2. Modify the sketch. You can resketch or redimension, but you cannot add notes, symbols, or draft entities.
3. Save the format sketch.

About Using Formats in a Drawing

Pro/ENGINEER saves drawing formats in a separate file. When you revise a format, it automatically updates it in all drawings that use it. When retrieving a drawing, if it cannot find the format that the drawing uses, it displays an error message in the message area.

For a multisheet drawing, you can have two default formats—one for the first sheet and another for the remaining sheets. The first sheet you create assumes by default the first sheet format. All of the remaining sheets use the second sheet format.

You can change the format on any sheet (including the first sheet) independently of the formats on other sheets; therefore, you could use a different format on each sheet of the drawing. To add or replace a single format on all existing sheets of a drawing, you must add the format to each individual sheet.

Each drawing format that you create has its own setup file—one that is completely independent of the drawing setup file. The configuration file option `format_setup_file` assigns a specified setup file to each format that you create, but does not do so for drawings. For these two setup files to have the same values, you must edit them separately. For more information, click *See Also*.

To Add or Replace a Format in an Existing Drawing

1. Using **Window > Activate**, set the drawing in which you want to add or replace a format as the active drawing.
2. On the DRAWING menu, click **Sheets > Format > Add/Replace**. The **Open** dialog box opens and displays the contents of the **System Formats** directory on your local system.
3. Select the format file you want to add or replace. Format file names contain the extension .frm. Sketched format files use the extension .sec.

To Add a Format to a Drawing While Creating the Drawing

1. Click **File > New... > Drawing > OK**. The **New Drawing** dialog box opens.
2. Under **Specify Template**, click **Empty with format**.
3. Under **Format**, select a format name by doing one of the following:
   – Choose a name from the list in the **Format** box.
   – Type [?] in the **Format** box and press ENTER; then select a name from the **Open** dialog box.
   – Click **Browse**, and then select a name from the **Open** dialog box.
4. Click **OK**. The system creates the new drawing and includes the specified format.

To Remove a Format from a Drawing

1. Using **Window > Activate**, make sure that the drawing from which you want to remove the format is the active drawing.
2. On the DRAWING menu, click **Sheets > Format > Remove**. The DWG SIZE TYPE menu and the DWG SIZE menus appear.
3. Assign a new format to the drawing by doing the following:
   – (Optional) Select a new drawing size from the DWG SIZE TYPE menu.
   – On the DWG SIZE menu, select a new drawing format size to use in the drawing.

   The system removes the current format and adds the newly specified format to the drawing.

To Blank or Unblank a Format on a Drawing

1. Using **Window > Activate**, make sure that the drawing from which you want to remove the format is the active drawing.
2. On the DRAWING menu, click **Sheets > Format > Blank** or **Sheets > Format > Unblank**.

   The format is blanked from the drawing (or redisplayed on the drawing if you selected **Unblank**).

To Display a List of Available Formats

1. Using **Window > Activate**, make sure that the drawing from which you want to remove the format is the active drawing.
2. On the DRAWING menu, click **Sheets > Format > List**.

   The system displays an Information window with a list of formats that are available in the current directory.

To Create Sheet Templates for Process Assembly Drawings

When working with a drawing of a Pro/PROCESS for ASSEMBLIES or Pro/PROCESS for COMPONENTS model, you can use the following procedure to create drawing sheet templates:

1. In the DRAWING menu, click **Sheets > Copy Process** to create a template from the current sheet. The **Process State** dialog box opens.
2. Choose from the options in the dialog box, and then click **OK**. The system adds a sheet to the drawing using the current sheet as the template.

The new sheet contains copies of the views and tables that were on the template sheet, and they reference the step that was highlighted when you clicked **OK**. The views retain their orientation.

Guidelines for Creating Drawing Sheet Templates

The following rules apply for creating sheet templates:

- You cannot use a sheet as a template if it contains views of more than one model.
- The **Copy Process** command appears in the SHEETS menu only if you are working with a drawing of a process assembly and you have a license for Pro/PROCESS for ASSEMBLIES or Pro/PROCESS for COMPONENTS.

To Set Up a Format Library

You use the configuration option `pro_format_dir` to set a path name (directory) where your company formats will be stored. When you use this configuration option, Pro/ENGINEER automatically searches the specified directory for company formats when adding or replacing formats in your drawing and layout. Pro/ENGINEER places modified formats in this directory when you save them.

1. Click **Utilities > Options**. The **Options** dialog box opens.
2. Enter the configuration file option `pro_format_dir`. This option uses a path name (directory) as its value, so you can create a single set of formats that everyone on the system can use, and place them all in a single directory.
3. Under **Value**, enter the path name and directory where you want to store all formats.

To Retrieve a Format from the Format Library

Using the **File Open** dialog box, you can retrieve formats from a format library directory within Pro/ENGINEER. To retrieve a format:

1. Click **File > Open**. The **File Open** dialog box opens.
2. Select **System Formats** from the **Look In** list. Navigate the menu tree until you locate the format.
3. Click **Open**. The selected format opens.

To Create a Drawing

When creating a drawing, you can specify a model in which to automatically place views.

1. Click **File** > **New**.
2. In the **New** dialog box, click **Drawing** and type a drawing name in the **Name** box (or use the default); then click **OK**. The **New Drawing** dialog box opens.
3. In the **Default Model** box, type the name of a model that resides in the current directory to use as the default. When you specify the optional default model, the system automatically brings in a format with parametric callouts (for example, &model_name) because the drawing searches the model for the parameter names. You can also type [?] or click **Browse...** to select a name from the **Open** dialog box. The default name is the last current model. Click **OK** when you are done.
4. Under **Specify Template**, do one of the following:
   - To use a Pro/ENGINEER drawing template, click **Use Template**, and then select a template from the list.
   - To create a drawing without a template but with an existing format, click **Empty with format**. Under **Format**, specify the format you want to use.
   - Specify the drawing size or retrieve a format. To specify the size, do one of the following:
     Click **Portrait** or **Landscape** in the **Orientation** box and select a standard size from the **Standard Size** list. **Portrait** makes the height larger than the width, and **Landscape** makes the width larger than the height.
     Or
     Click **Variable** in the **Orientation** box to define both the height and width dimensions. Select **Inches**

or **Millimeters** and type values in the **Width** and **Height** boxes.
To retrieve a format, select **Retrieve Format** and select a name from the **Name** list in the **Format** box. You can also type [?] or click **Browse** to select a name from the **Open** dialog box.
5. Click **OK**. The system displays the drawing as specified and the DRAWING menu appears.

To Add a Format to a Drawing

1. Choose DRAWING > **Sheets** > **Format** > **Add/Replace**.
2. In the **Open** dialog box, select a previously created drawing format residing in the System Formats directory, or type [?] to select from a menu of previously created formats. To select from a namelist:
   – Under **Look In**, Choose the name of another directory to access formats in other locations.
   The system adds the format to the drawing, with the lower-left corner of the format coinciding with the lower-left corner of the drawing sheet.

Importing Drawing Formats

You can also import drawing formats through IGES, DXF, TIFF, CGM, or SET.

You can add the following items to your drawing in any order you require: drawing formats, draft geometry, and model views. For more information, refer to the topics , Before and After Adding Views, and To Add a Format to a Drawing.

Before and After Adding Views

Before you start adding views, you must add the first model to the drawing (if you did not already do so during creation of the drawing by specifying the optional default model. For more information, refer to the topic To Create a Drawing). If you retrieve a model that has an instance, you must specify whether you want to retrieve the instance or the generic model. You can use a single model to create any number of drawings. When creating each new drawing, give the drawing a unique name.

After you add the first view, the information at the bottom of the main window changes according to that view. Pro/ENGINEER sets the scale so that several views can fit on the drawing. You can modify this scale any time after you add the first view.

To Add the First Model to a Drawing

If you did not specify a model to which the drawing refers during creation of the drawing, you can use the following procedure to add the first model to the empty drawing.

1. Choose DRAWING > **Views**. The **Open** dialog box opens.
2. In the **Name** box, type the name of the model you want to add, or locate the desired model using the **Look In** box. Click **Open** when you are done.
3. Add the first view by choosing VIEWS > **Add View**; then select view specifications from the VIEW TYPE menu.

Retrieving Drawings

Using the **File Open** dialog box (click **File > Open** to access the dialog), you can retrieve an existing drawing by selecting it from a list of drawings in the current directory. The list includes the names of all directories contained in the current directory and buttons to go up to the next directory level. You can also use the **Look In** box to navigate to the directory where the drawing resides. If you click the **Working Directory** button, the system lists all drawings in the working directory. When you retrieve the drawing, it retrieves the associated model(s) and regenerates the drawing.

To Retrieve a Model into the Current Session

To open a model from within Drawing mode, you can use **File > Open**, or you can use the Model Tree.

1. Click **File > Open**. The **File Open** dialog box opens.
2. Navigate to and select the model you want to open, and then click **Open**.

The model is retrieved into the current session in a separate Pro/ENGINEER window.

Using the Model Tree

1. In the Model Tree, select the model you want to retrieve, and then right-click to display the shortcut menu.
2. Choose **Open** from the shortcut menu. The model is retrieved into the current session.

To Copy Drawing Files With Object Rename

Using the configuration file option `rename_drawings_with_object`, you can copy a drawing and a part simultaneously with its associated part or assembly file (using **File > Save As**) and then rename the drawing. The drawing adopts the same new name as the name of the copied part or assembly file.

1. Click **Utilities > Options**. The **Options** dialog box opens.
2. Type the configuration option `rename_drawings_with_object` in the **Option** box.
3. Select a value for the option from the **Value** drop-down list.

    When you specify the value as `both`, `part`, or `assembly`, and then choose **Save a Copy** from the Pro/ENGINEER **File** menu for a part, the system also renames the drawing file associated with the renamed part, as long as it has the same name as the part.

To disable this option, set this configuration option to `none` (the default setting).

The following diagram illustrates the renaming process when you use **Save As** to save a part or assembly file that has an associated drawing.
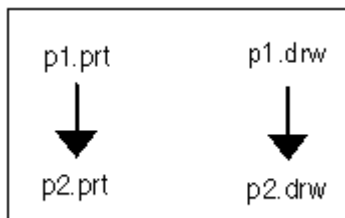
About Adding Views

The first view that Pro/ENGINEER adds to a drawing is a general view. Initially, it displays it on the sheet in the default orientation, but you can orient it using the VIEW TYPE menu (accessed by clicking DRAWING > **Views > Modify View > View Type**). For each view, you must specify the following properties:

*   The type of view, or how it is created (such as oriented independently of other views, projected from an existing view)
*   How much of the view is shown
*   If the view has a cross section
*   If the view is scaled, or if you want to create a perspective view

The next table summarizes valid menu combinations of these view types.

Valid View Type Menu Combinations

	Projection	Auxiliary	General	Detailed	Revolved	Graph
Full View	X	X	X	X	X	X
Half View	X	X	X			

Broken View	X		X			
Partial View	X	X	X		X	
Section	X	X	X		X	
No Xsec	X	X	X			
Of Surface	X	X	X			
Scale			X	X		X
No Scale	X	X	X		X	X

Note: You can also use the **Orientation** dialog box (accessed by clicking DRAWING > **Views > Modify View > Reorient**). However, this dialog box contains limited options for reorienting a view.

Draft Views

Using the **Draft View** command (DRAWING **> Tools > Draft View**), you can set a drawing view to be the current draft view so that Pro/ENGINEER associates all new draft entities with that view. When you have associated draft entities with a drawing view, they move with the view when you move it, maintaining their location relative to that view. Also, when you scale the view or the drawing, the system scales all draft entities associated with a view by the same factor.

The system uses the view scale of the current view when you create draft entities.

When a view is set to be current (using DRAWING > **Tools** > **Draft View** > **Set Current**), the system uses the view scale of the current view for draft entities that you sketch, such as lines and circles, as well as for dimensions related to draft entities.

The current draft view scale is used only when the current view is on the current sheet of the drawing.

Using the current draft view illustrates the associativity between draft entities and the current draft view. You can associate notes without leaders, symbols, and geometric tolerances with a draft view in addition to draft geometry. However, you *cannot* associate draft datums and datums with a draft view.

Controlling How a Drawing View Is Displayed

The configuration file option `display_in_adding_view` controls how Pro/ENGINEER displays a model in a drawing. If you set this option to `wireframe`, the system adds each new view in wireframe and displays datums. If you set it to `minimal_wireframe`, it initially displays each new view in wireframe without datums, axes, or silhouette edges. You can also use the default view display to add views with or without hidden lines.

To Create a General View

1. On the DRAWING menu, click **Views > Add View > General**, and then accept the default commands **Full View**, **No Xsec**, and **No Scale**. Click **Done**.
2. Select a location on the drawing as the center point for the new view. The view appears centered in the selected location. The **Orientation** dialog box opens.
3. Orient the view using commands in the **Orientation** dialog box. If you want to retrieve a previously saved

view of the model in that orientation, click **Saved Views** and select the view name.

4. After you have specified an orientation, click **OK** to close the **Orientation** dialog box; then, click **Done/Return** on the VIEWS menu.

When you create a projection view and move the general view, the projection view follows the main view to the new location.

Example: Creating and Orienting the General View
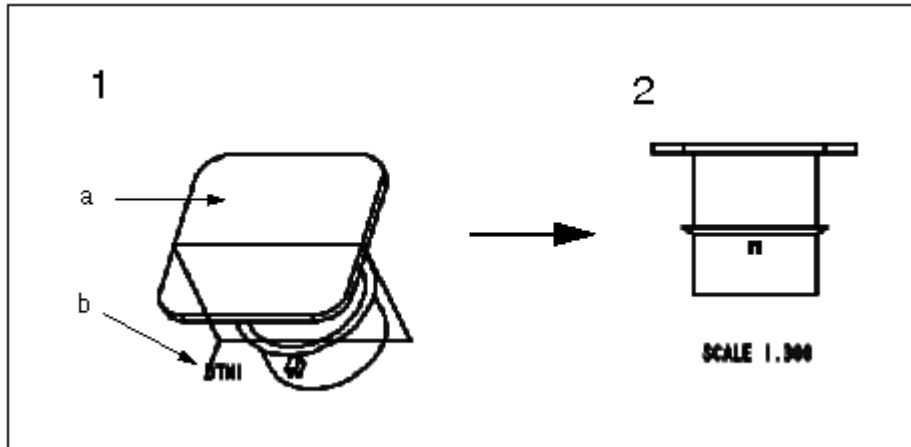
1 The general view appears in default orientation with datums shown.
a Select this surface for the Top.

b Select this datum for the Front.

2 The system orients the view as specified. If **Display Datum Planes** is selected in the **Environment** dialog box, datums perpendicular to the screen remain visible.

To Set a Drawing View as the Current Draft View

1. Click DRAWING **> Tools > Draft View > Set Current**.
2. Select a view to be current. The system sets the selected view as the current view.
3. To add existing draft entities to the current draft view, choose **Add** on the DRAFT VIEW menu, and then select the entities you want to add. To associate the entities with the view, you must add the draft entities to the view first and then dimension them. If you have previously associated an entity with another view, the system automatically removes it (disassociates) from the original view when you add it to this view.
   – To delete the draft entities, click **Remove**, and then select the entities you want to remove from the current view. Once you remove them, the system disassociates those entities from the view.
   – **Note: Add** and **Remove** are available *only* when the current view is set.
   – To unset the current draft view, leaving no view set as current, use **Unset Cur**.

About Working with Model Datum Planes

Pro/ENGINEER shows datum planes in the drawing under these conditions:

- Temporarily, when you are orienting the model while creating a drawing view (this is the *only* time it displays a datum plane that is not perpendicular to the screen).
- When the datum is a draft datum. When you have selected **Display Datum Planes** in the **Environment** dialog box (or when you have selected **Planes** in the **Datum Display** dialog box) and the datum planes are perpendicular to the screen.
- When you have set the datum plane as a reference datum and it is perpendicular to the screen.

To manipulate set datums, use **Edit > Properties** on the menu bar. Set datums appear regardless of the setting

in the **Environment** or **Datum Display** dialog boxes. You can erase them from a view by choosing **View > Show and Erase** on the menu bar, or by blanking them individually or on a layer.

You can also use shortcut menus (accessed by clicking the right mouse button on an object) to modify and manipulate 3-D set datums in the following ways:

- Move them to a different location on the drawing sheet.
- Redefine them.
- Erase and unerase them.

To Create a Datum Plane Feature

In Drawing mode, you can create datum plane features in a model; however, you cannot create a datum plane on the model if the model is read-only in the drawing.

1. Choose **Insert** > **Datum > Plane**.
2. In the **Datum** dialog box, type a name in the **Name** box. If you do not specify a name, the system assigns the name as DTM#.
3. Do one of the following:
   - Click **Define**, and then constrain the datum by choosing commands from the DATUM PLANE menu; then click **Done**.
   - Click **On Surface** to place the datum plane feature on a planar model surface.
4. Under **Type**, choose the type of datum you want to create.
5. If you are creating a set datum, click **Free** or **In Dim** to specify its placement.
   When the system has fully constrained the datum, it creates it; however, it displays the datum *only* if it is perpendicular to the screen and you have selected **Display Datum Planes** in the Environment dialog box (or if you have selected **Planes** in the **Datum Display** dialog box).
6. To continue creating datums, click **New**, type another name in the **Name** box, and click **Define...** or **On Surface...**.

To Rename a Model Datum

1. Click **Edit > Properties**.
2. In the drawing window, select the datum you want to rename. The **Datum** dialog box opens.
3. Type a new name in the **Name** box.
4. Click **OK** to save the name change and to close the dialog box.

To Modify the Display of a Set Datum Plane

You can lengthen or shorten set datum planes by moving or clipping them. Moving a datum plane moves either side of the datum. Clipping moves the end opposite to the datum name.

1. Select the datum you want to modify. The datum is highlighted and its handles display.
2. Do one of the following:
   - To move the datum, select anywhere in the drawing window and the datum text moves to a corresponding location along the datum. The name of the datum also flips to the side of the datum where you selected.
   - To clip the datum, select anywhere and the datum plane moves to a corresponding point along the datum. The datum name does not move.

To Show Datum Planes

Datum planes that you have already set as reference datums appear automatically in those views where the datums are perpendicular to the screen (on edge). If you have erased a set datum plane, use this procedure to show it again.

1. Click **View > Show and Erase**.
2. In the **Show/Erase** dialog box, click **Show**.

3. Under **Type**, click the **Datum Plane** button.
4. Click **OK** when you are done. The system displays all datum planes.

To Erase a Set Datum

1. On the Pro/ENGINEER menu bar, click **View > Show and Erase**.
2. In the **Show/Erase** dialog box, click **Erase**.
3. Click the **Datum Plane** button in the **Type** box; then select a button in the **Erase By** box to specify the feature, view, or part.

To restore it, click **Show**, the **Datum Plane** button in the **Type** box, and a button in the **Show By** box to specify the feature, view, or part.

Erasing a Set Datum from a Member of the Assembly

If a drawing has several occurrences of the same part in an assembly, you can erase a set datum from a particular member of the assembly without causing the selected set datum to disappear from other occurrences of the member.

Note: You can erase only set datum planes from the drawing. To remove from display any other model datum planes (not draft datums), clear **Display Datum Planes** from the **Environment** dialog box, or clear **Planes** from the **Datum Display** dialog box (**View > Datum Display**).

To Create a Draft Datum Plane

1. On the Pro/ENGINEER menu bar, click **Insert > Draft Plane**. The GET POINT menu opens in the Menu Manager, and **Pick Pnt** is selected by default.
2. Click the left mouse button to begin drawing the datum plane.
3. Extend the datum plane to the proper orientation. Place the endpoint by clicking the left mouse button.
4. Type the name of the datum plane.

Note: To move or reorient draft datums, select a datum and then click anywhere in the drawing window to relocate or clip it. You can also click DRAWING > **Tools > Rotate** or DRAWING > **Tools > Translate**.

Displaying Datum Planes

Pro/ENGINEER shows datum planes on edge, making both red and yellow sides visible. You can move, delete, and restore draft datums using the shortcut menu (accessed by right-clicking on the datum).

To Control the Size and Shape of Datum Points

To control the size of model datum points and sketched datum points in Drawing mode, use the following drawing setup file options:

- Specify a value for the drawing setup file option `datum_point_size`. To open the drawing setup file, click **Advanced > Draw Setup** on the DRAWING menu. The default size is `.3125`.

- To display draft or datum points as a cross, dot, circle, triangle, or square, specify a value for the drawing setup file option `datum_point_shape`.

Note: These drawing setup file options control the display of points in Drawing mode only. To modify the datum point symbol display for Part or Assembly mode, set the configuration file option `datum_point_symbol`.

About Working with Model Axes

You can manipulate the axes of cylindrical or conical surfaces independently of the **Environment** dialog box settings. However, keep in mind the following:

- The system shows the axis as a line whenever you view it from the side, or as a cross hair whenever you orient the axis normal to the screen.
- In Drawing mode, if you have not cleared **Display Datum Axes** from the **Environment** dialog box, axis names do not appear on the drawing. You can control the display of axis names independently of the axis display by selecting or clearing **Axis Tags** on the **Datum Display** dialog box (**View > Datum Display**).
- You can create axes in the drawing if you have a Pro/DETAIL license; however, these axes belong to the drawing, not to the model.
- A draft axis always appears as a centerline.

By right-clicking a selected axis and using the shortcut menu, you can modify set and unset model axes in the following ways:

- Move them to a different location on the drawing sheet.
- Modify their attachment.
- Redefine them.
- Erase and unerase them.

To Create a Datum Axis

The system displays draft axes in leader style with centerline font.

1. On the Pro/ENGINEER menu bar, click **Insert > Datum > Axis**.
2. In the **Axis** dialog box, type a name in the **Name** box.
3. Click **Define...**.
4. Constrain the datum axis by choosing commands from the DATUM AXIS menu; then choose **Done**.
5. To continue creating datum axes, click **New**, type another name in the **Name** box, and click **Define...**.

To Rename a Datum Axis

1. In the drawing window, select the datum axis you want to rename, and then right-click and select **Redefine** from the shortcut menu.
2. In the **Axis** dialog box, type a new name in the **Name** box.
3. Click **OK** to close the dialog box. The axis is renamed.

To Create a Set Datum Axis

1. In the drawing window, select the datum axis on the screen, and then right-click and select **Redefine** from the shortcut menu.
2. In the **Axis** dialog box, click -**A-** (set). The system encloses the axis in a feature control frame.
3. Click **OK** to close the **Axis** dialog box.

To Specify the Placement of a Set Datum Axis

1. In the drawing window, select the axis you want to place or relocate, and then right-click and select **Redefine** from the shortcut menu.
2. Click **In Dim** and then **Pick Dim** in the **Axis** dialog box.
3. Select a dimension in the model. The system places the set axis under the dimension on the screen.
4. Click **OK** to close the dialog box and save the change.

Note: If you set the drawing setup file option gtol_datums to STD_ASME or STD_ISO, you can also place a set datum axis on a geometric tolerance. Click **In Gtol** and **Pick Gtol...**; then select a gtol in the model.

Tip: How the Show and Erase Command Affects Axis Display

The **Show and Erase** command in the **View** menu (menu bar) displays the axes of cylindrical and conical surfaces on the drawing and erases axes that the system shows in a drawing. Because axes belong to the model,

like dimensions, you cannot delete them.

Note: Showing or erasing an axis in a radial pattern affects corresponding axes in all pattern instances.

To Display Axes on a Drawing

1. On the Pro/ENGINEER menu bar, click **View > Show and Erase**.
2. In the **Show/Erase** dialog box, click **Show**.
3. Click the **Axis** button in the **Type** box; then select a button in the **Show By** box to specify the feature, view, or part. If you select **Feat_View** or **Part_View**, select the feature or part, choose **Done Sel**, and then select the view in which to show the axis.

Controlling the Display of Axes

By setting the drawing setup file option `axis_interior_clipping` to `no`, you can set the axis display according to these three ANSI Y14.2M standard requirements:

- Line starts and ends with a dash.
- Perpendicular axes of a centerline intersect at a short dash.
- Very short centerlines are unbroken.

Axes that are set to CTRLFONT actually are shown as CTRLFONT_S_L or CTRLFONT_L_L, depending on their orientation. You can control them by setting `line_style_length` to CTRLFONT_S_L or CTRLFONT_L_L, not CTRLFONT. To control axes that are perpendicular to the view, set `line_style_length` to CTRLFONT_S_L; to control axes that are parallel to the view, set it to CTRLFONT_L_L.

This option affects the axis display in the following ways:

- Axes parallel to the screen appear in one line in a centerline font, beginning and ending with a long dash; choose **Move** from the **Edit** menu (menu bar) to modify either endpoint.
- Axes perpendicular to the screen appear in two lines, each of them drawn in two segments.
- Perpendicular lines intersect at a short dash.
- You can only control endpoints using the **Move** command (the **Clip** command is not available for perpendicular axes).
- You cannot move an endpoint of a perpendicular axis beyond the axis center.
- When any axis is too short, it appears in a solid line font.

Using the drawing setup file option `radial_pattern_axis_circle`, you can set the display mode for axes of rotation that are perpendicular to the screen in radial pattern features. When you set the option to `yes`, a circular shared axis appears, and the axis lines pass through the center of a rotational pattern (see Appendix A, Drawing Setup File Options, for details). However, this only affects patterns that you create using the **Dim Pattern** command. The following restrictions apply when you set this drawing setup file option to `yes`:

- You cannot move, clip, or modify the line style of a circular shared axis that appears once you choose the **Show** command button from the **Show/Erase** dialog box (accessed by clicking **View > Show and Erase** on the menu bar).
- You cannot use the **Mod Attach** command on axis lines or circles of patterned features.
- In a clipped view, the system only displays portions of lines and circles that are inside a view boundary.
- You cannot use radial patterns of group features.
- You cannot show radial pattern axis circles for reference patterns.

To Modify the Line Style of a Model or Draft Axis

1. On the Pro/ENGINEER menu bar, click **Format > Line Styles**. The LINE STYLES and GET SELECT menus open in the Menu Manager.
2. Select the axis name or the axis itself (for model axis features). The **Line Style** dialog box opens.
3. In the **Line Style** dialog box, select a line style from the **Style** list.

4. Click **Apply**. The axis acquires the specified line style. To reset it to the old style, click **Reset** and **Apply**.
   Note: When you set the drawing setup file option `axis_interior_clipping` to `no`, you can still change the axis line style. However, this disables the automatic adjustment of a pattern.

To Modify Axis Length

To extend or shorten the axis lines, move or clip the lines. Moving the lines moves whichever end you select. Clipping the lines moves the portion of the axis within the circle closer or farther from the origin.

To Move an Axis, when the Axis is parallel to the screen

In the drawing window, click on the axis to select it, and then pick on either endpoint (handle) and drag the axis to its new location.

To Extend or Clip Axis Lines when the Axis is Perpendicular (normal) to the Screen

- Click on the axis to select it. The crosshairs, endpoints (handles), and axis name highlight. Then do one of the following:
    - To clip or extend only a portion of the axis, select and drag a single handle (endpoint).
    - To clip or extend all axis lines at same time, select and drag the axis name.

To Create a Break in a Model Axis Line

1. In the Graphics window, select the axis line in which you want to create the break.
2. On the Pro/ENGINEER menu bar, click **Insert > Break**. The BREAK menu opens in the Menu Manager. **Add** is the default selection.
3. Select the first location on the axis line to begin the break.
4. Select the second location to finish the break. The system creates the break.

To Delete a Portion of a Normal-to-Screen Axis

1. Select an axis line, and then click **Edit > Delete**.
2. To resume the display of all portions of the axis, click **View > Show and Erase** The **Show and Erase** dialog box opens.
3. Click **Axis** in the **Type** box.
4. Click **Show All**. The deleted axis lines are redisplayed and highlighted in a different color. If **Preview** is selected in the dialog box, click **Accept All**. All axis lines are restored to the display.
   Note: If you choose the normal-to-screen axis, the system erases all of the lines for that axis.

To Create an Axis Symmetry Line

Using the following procedure, you can create an extension between axis cross hairs showing that the axes are symmetrical about a dimension.

1. On the menu bar, click **Insert > Symmetry Line Axis**.
2. Select two axes that are perpendicular (normal) to the plane of the screen (shown as cross hairs). To extend the symmetry line to either side, grab a handle of the axis with the mouse pointer and click anywhere outside of the line.

Example: Axis Symmetry Lines
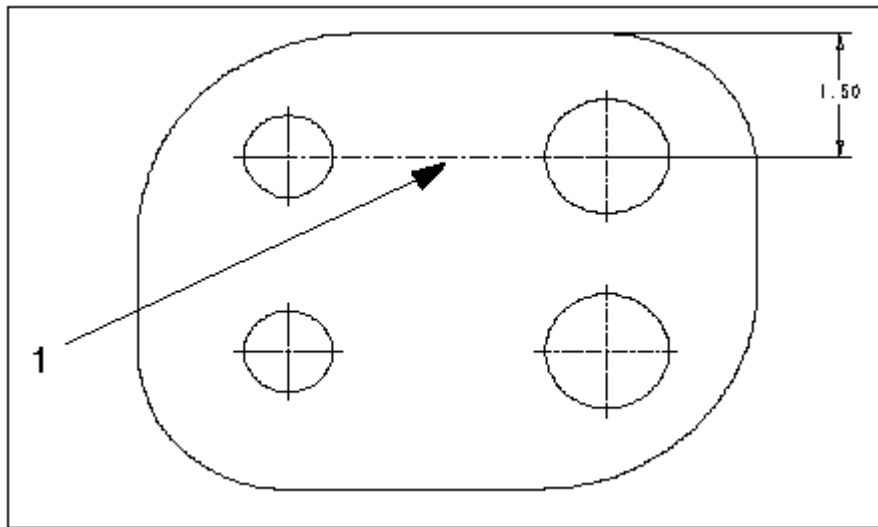
1. Axis symmetry line.

About Rotating Axes

Using the **Mod Attach** option in the right mouse button shortcut menu, you can rotate model axes that are normal to the screen. When you rotate the axis, it rotates according to another reference entity that you select, such as an edge or point, as shown in the following diagram. The rotated axis remains normal (perpendicular) to the screen; however, the cross reorients according to your specifications. To open the shortcut menu and access the **Mod Attach** option, select and then right-click on the axis.

Rotating Axes
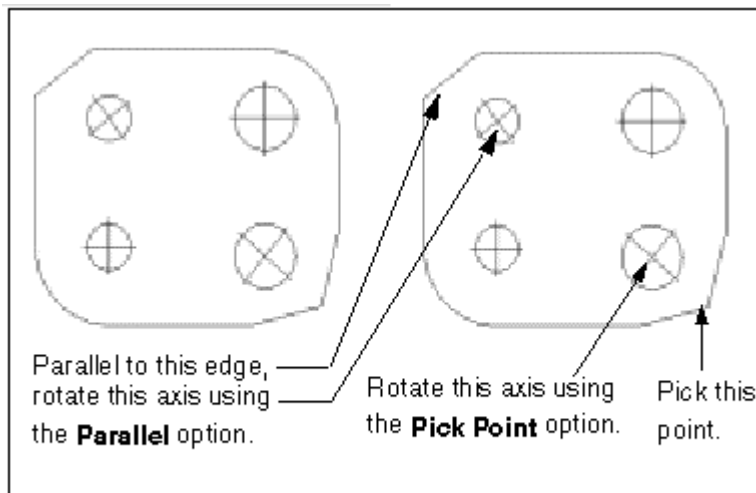
The **Rotate Axis** menu displays the following options:

- **Through Geom**—Picks an edge, datum point, or center of an axis normal to the screen through which the axis line that is closest to that point passes.
- **Pick Point**—Picks a point on the screen through which the axis line passes.
- **Parallel**—Picks a linear edge or datum curve to which the axis line is parallel.

- **Horizontal**—Returns the axis to its standard orientation.
- **Enter Angle**—Specifies an angle for axis rotation about the X-axis.

To Rotate an Axis

1. Select the axis you want to rotate, and then right-click on the axis to display the shortcut menu.
2. Click **Mod Attach** in the shortcut menu.
3. Select another axis center or edge as a rotation reference to orient the selected axis, and then press ENTER. The axis rotates in reference to the edge or second axis reference.

To Create a Draft Axis

1. On the Pro/ENGINEER menu bar, click **Insert > Draft Axis**.
2. Click the left mouse button to start drawing the axis.
3. Move the mouse pointer to extend the axis to the proper orientation (the axis line drags with the mouse). Place the endpoint by clicking the left mouse button.
4. Type the name of the axis. The system displays the name.
5. To move or reorient the axis, you can move or clip it by selecting the axis and then clicking anywhere in the drawing window, or choose **Rotate** or **Translate** from the DRAWING > TOOLS menu. You can also use the right mouse button shortcut menu, as described in the topic About Using the Shortcut Menu.

About Using the Shortcut Menu

Using a shortcut menu or the **Edit** menu (menu bar), you can modify any object in a drawing from anywhere in the menu tree. At any time when a drawing window is active, you can interrupt your current process and activate a drawing object for modification. When you finish making modifications, the system returns you to your original location in the menu tree.

Manipulating Snap Lines

Using the shortcut menu, you can modify and manipulate snap lines in the following ways:

- Change the spacing.
- Change the attachment.
- Delete snap line ends.
- Move snap line ends.

Modifying Items Using the Shortcut Menu

When using the right mouse button shortcut menu or the **Edit** menu (menu bar) to modify a detail item, keep in mind the following:

- If you move a note in a table by choosing **Move** from the **Edit** menu or **Move/Activate** from the shortcut menu, the system moves the table.
- You cannot modify unset 3-D datums.
- When you are removing dimensions, the same restrictions apply to the **Delete** commands in the Edit menu. For example, you cannot delete dimensions that belong to features.

Using the Box Handles

As in Sketcher mode, you can use the box handles to manipulate entities in a variety of ways, including the following:

- Rubberband the radius of a draft circle by selecting one of the handles.
- Drag the endpoint of a line by using one of the handles.
- Select a handle on the minor or major axis of an ellipse and then rotate and resize the centerpoint while leaving the other axis fixed.

- Move the text of a dimension that does not have an elbow.
- Move one endpoint of an arc, while leaving the other one fixed.

To Modify an Item Using the Shortcut Menu

1. Select an item in the drawing window. The item is highlighted and handles appear to emphasize the portions of the item that you can move. The pointer changes as you move over the handles. Then, press the right mouse button. A shortcut menu appears with options for modifying the selected item. The options on the shortcut menu vary depending on the item you have selected. For example, if you select a witness line, the shortcut menu displays the options **Erase Wit Line** and **Default Wit Line**, among other appropriate options.
Note: You may need to press the right mouse button for about one-half of a second before the pop-up menu appears.
Note: At any time during this process, you can also use the right mouse button to display the pop-up menu again and choose a command from that menu.
2. Edit the selected item as necessary or appropriate. To deactivate the selected object, click anywhere in the drawing window or select another object. With some actions, such as exploding a symbol, the system deactivates the active item automatically because the object no longer exists in the same form.

About the Major View Types

You access the VIEW TYPE menu by choosing **DRAWING > Views > Add View** on the Menu Manager. Using the VIEW TYPE menu, you can specify eight major view types, as well as how much of the model to show in a view, whether the view is of a single surface or has cross sections, and how the view is scaled.

- A *general* view is a view that is independent from other views in the drawing, and shown in the default orientation specified in the Pro/ENGINEER environment.
- A *detailed* view is a portion of a model shown in another view. Its orientation is the same as the view from which it is created, but its scale may be different so that you can better visualize the portion of the model that you are creating. The display of edges in a detailed view follows that of the view from which it is created (its *parent* view).
- A *projection* view is an orthographic projection of another view's geometry along a horizontal or vertical direction. You can specify the projection type in the drawing setup file by basing it on third angle (default) or first angle rules.
- An *auxiliary* view is a projection of the geometry of another view at right angles to a selected surface or along an axis. The selected surface in the parent view must be perpendicular to the plane of the screen.
- A *revolved* view is a planar area cross section from an existing view, revolved 90 degrees around the cutting plane projection, and offset along its length. It can be full, partial, exploded, or unexploded.
  Note: Detailed, projection, auxiliary, and revolved views have the same representation and explosion offsets, if any, as their parent views. You can simplify, restore, and modify the explosion distance of each view without affecting the parent view. However, detailed views *always* appear with the same explosion distances and geometry as their parent views.
- A *graph* view shows the sketch of a graph feature and its dimensions. The system updates any changes parametrically.
- An *of flat ply* view is a flat single-ply view of a composite model. It can exist in a regular drawing or in sequence drawings.
- A *copy & align* view is an aligned partial view based on a specified view boundary and an alignment relative to the existing partial view.

To Create a Drawing View Using Saved Part and Assembly Views

In Drawing mode, you can recall views saved as named view orientations in Part and Assembly modes and use them to create a drawing view.

1. Choose **DRAWING > Views > Add View** on the Menu Manager.
2. Add a general view. When you place the view on the screen, the **Orientation** dialog box opens.
   Note: The recalled view uses the default view as a starting point for its orientation. If you have selected a different default view type in the **Environment** dialog box since the view was created, it does not look the same. If this occurs, change the default view. Select a name from the **Saved Views** list in the **Orientation** dialog box when reorienting the general view.
3. Select the name of the saved view from the list, and then click **Set**. The view being created assumes the orientation of the saved view. Its scale, however, does not change.
4. Click **OK** to close the **Orientation** dialog box.

General Views

The first view that you add to a drawing is a general view. You add other general views in the same way that you add the first one. The system places them on the screen in the default orientation. If you reorient a general view that has dependent views in the drawing, the system reorients those views as well. You can move general views anywhere on the drawing and scale them, if necessary.

Changing the Default Orientation

A model appears on the screen in its default orientation if you set the configuration file option `disp_trimetric_dwg_mode_view` to `yes`. If you set it to `no`, a model does not appear until you choose **Default** from the **Orientation** dialog box. However, when you create a drawing view using the default orientation, if you change the setting of this configuration file option, the system does not change the orientation. Once you set a view in the default orientation, the system freezes it.

For example, if you place a general view with default orientation and select **Trimetric** from the **Default Orient** list in the **Environment** dialog box, the system does not reorient the view if you select **Isometric** or retrieve the drawing in an isometric environment. To reorient the view, choose **Views > Modify View > Reorient** from the DRAWING menu (Menu Manager). Select a general view to reorient, and then, in the **Saved Views** section of the **Orientation** dialog box, select **Default**.

When changing the default orientation of a drawing view, keep in mind the following:

- If you delete or suppress geometry that the system is using to orient a view, it changes that view and all of its children to the default orientation. You cannot recover the original view orientation if the geometry was deleted. Resuming the suppressed feature, however, restores the original orientation of the view.

- The system freezes the orientation of default views in any pre-Release 18.0 drawings after you save them in Release 18.0.

- Before placing a cross section defined as planar in the view, you must orient the view so that the plane of the section is parallel to the plane of the screen. If the cross section is revolved, the plane of the section must be perpendicular to the screen.
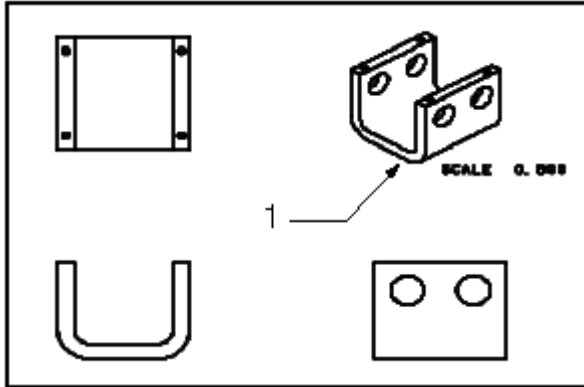
1   A general view with scale factor.

To Add a General View

The procedure for adding a general view to a drawing differs slightly depending on whether an active model is open in the background.

If the Model is Active:

1. On the DRAWING menu, choose **Views > Add View**.
2. From the VIEW TYPE menu, select the type of view you want (the default is general), and then click **Done**.
3. Place the view onto the drawing sheet by clicking a location on the sheet. The active model displays in 3-D orientation, in isometric view. The **Orientation** dialog box opens and prompts you to orient the model.
4. Select orientation references. The model reorients automatically. Then, click **OK**.

If the Model is Not Active

1. Choose **Views > Add View**. The system prompts you to select a model and opens the **Open** dialog box.
2. Select a model in which to place the view, and then click **Open**. Once the model is open, you can follow steps 2 through 4 in the preceding procedure.

To Reorient a View

You can reorient a view, including changing the view angle, by using the following procedure.

1. Choose DRAWING > **Views > Modify View** > **Reorient,** and then select the view. The **Orientation** dialog box opens.
2. If the view has any dependent (*child*) views, the system highlights them and warns you that it is going to reorient them also. Type [Y] to continue.
3. Using the **Orientation** dialog box, follow the standard procedure to specify the view orientation.

Detailed Views

A detailed view displays a portion of the model shown in another view. To create one, you define a spline boundary around the portion of an existing view that you want to show in detail. As shown in the following figure, the system displays the view name and scale value below it, circles the portion of the parent view that is detailed, and attaches a note identifying the detailed view.

The display of a detailed view follows that of the view from which you created it. For example, if the parent view displays hidden lines in the area that is detailed, the detailed view (which is simply a portion of its parent) also displays those hidden lines. Also, if you erase a feature from the parent view, the system also erases it from the detailed view. Because of this dependency, you can modify such characteristics as cross-hatching and hidden line display in a detailed view *only if you modify the parent view as well.* However, you can make a detailed view independent of its parent.

Example: A Detailed View
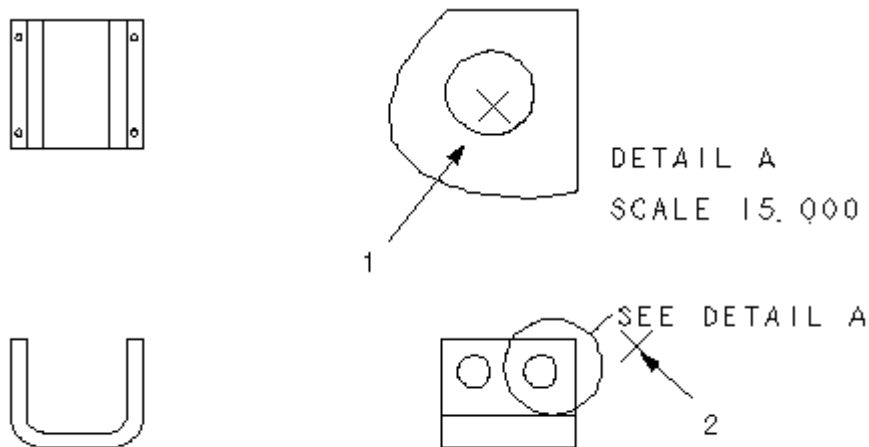
1   Placement pick.
2   Select note location.

To Add a Detailed View

1. On the DRAWING menu, choose **Views > Add View**. The VIEW TYPE menu appears.
2. Choose **Detailed** and other available commands from the VIEW TYPE menu.
3. Choose **Done** to accept the commands, or **Quit** to quit the creation of a new view.
4. Select the location of the new view on the drawing.
5. Type the scale value for the view.
6. Select a point on an edge in an existing view. The selected geometry is highlighted and a red cross appears, representing the reference point on the geometry to be in the detailed view. The system needs this reference point to properly regenerate the detailed view.
7. Enclose all of the geometry that you want to show in the detailed view. Sketch a spline around the red cross without intersecting other splines to define an outline. Press the middle mouse button when you are done.
8. Type a name for the detailed view (the system prefixes the name with the word "DETAIL"). Press ENTER.
9. From the BOUNDARY TYPE menu, choose **Circle**, **Ellipse**, **H/V Ellipse**, **Spline**, or **ASME 94 Circ**. Sketch a spline on the parent view that reflects the selected boundary type, and then press the middle mouse button to close the boundary. If the spline boundary does not enclose the red cross, you must resketch the boundary. The sketched spline and the new detail view display.
10. If you selected **Circle**, **Ellipse**, **H/V Ellipse**, or **Spline** as the boundary type, select the location for the detail note that is attached to the parent view.
11. If you selected **Ellipse**, **H/V Ellipse**, or **Spline** as the detailed view circle type, select the leader attachment point for the detailed view note. The system displays the view with its name and scale below it. A circle appears around the parent view approximating the size of the sketched boundary.
12. Make the following modifications, as necessary:
    –   Modify the size of the sketched boundary.
    –   To change the scale for the view, choose **Modify View** from the VIEW menu, then choose **Change Scale** from the VIEW MODIFY menu. Select the detail view, and then type a new scale and press ENTER. Erase the scale name using the **Show/Erase** dialog box (click **View > Show and Erase** on the menu bar).
    –   Edit the name of the detailed view by choosing **Modify View** on the VIEW menu, then **View Name** on the VIEW MODIFY menu. Select the detail view as the view you want to rename, and then type the new name (the view name can contain up to 80 characters).
    –   Change the display of the circle, leader, and note by modifying the setting of these drawing setup file

options:

- `detail_circle_line_style`
- `detail_circle_view_note`
- `detail_view_circle`
- `view_note`

– Make the display of the detailed view independent of its parent view by choosing **Disp Mode** from the VIEWS menu, then choose **View Disp** > **Det Indep**. To reset the dependency, choose **From Parent**.

– Use the **Move** command in the **Edit** menu (on the menu bar) to move the detailed view name along the **ASME 94** circle.

– Erase the detailed view note using the **Show/Erase** dialog box (**View > Show and Erase** on the menu bar).

Note: When you create a detailed view of a part containing an axis that lies off the part (that is, model geometry does not enclose it), the axis does not appear in the detailed view.

Projection Views

Pro/ENGINEER automatically creates projection views by looking to the left, right, above, and below the selected view location to determine the orientation. The arrows in the next figure show the direction the system looks to create projection views. When it finds conflicting view orientations, you must select a view to be the parent view. It then constructs a view from the selected view. If the system cannot find another view within its line of sight, you must select again at another location to place the projection.

A projection view acquires the scale of its parent view.

Examples: Adding Projected Views
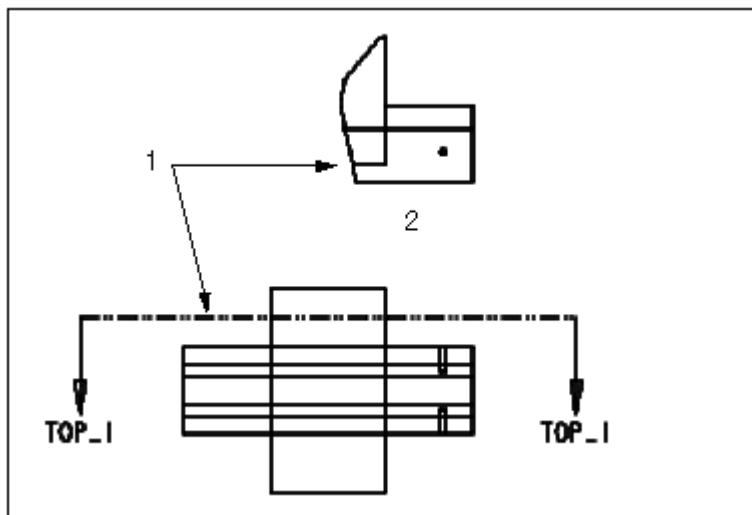
1   Arrows added for partial projection view: TOP_1".
2   VIEW TOP_1".

To Add a Projection View

1. On the DRAWING menu, choose **Views > Add View**. The VIEW TYPE menu opens.
2. Choose **Projection** (default selection) and other available commands from the VIEW TYPE menu.
3. Choose **Done** to accept the commands, or **Quit** to quit the creation of a new view.
4. Select the location for the new view on the drawing. The system displays the projection view.
5. Make the following modifications, as necessary.
    – Show or delete arrows that point back to the view from which you created the projection by choosing

Modify View from the VIEW menu, and then choosing **Add Arrows** or **Del Arrows** from the VIEW MODIFY menu.

- Automatically add view names to projection views in the format "VIEW viewname-viewname" by setting the configuration file option `make_proj_view_notes` to `yes`.

6. Change the view name or projection view arrow name by choosing VIEWS > **Modify View > View Name.** Then select the projection view and type a new view name.

Auxiliary Views

Auxiliary views discern the true size and shape of a planar surface on a part. The system makes a projection of the model perpendicular to a selected edge. You can create an auxiliary view from any other type of view.

To Add an Auxiliary View

1. Choose VIEWS > **Add View > Auxiliary** and other available commands from the VIEW TYPE menu.
2. Choose **Done** to accept the commands, or **Quit** to quit the creation of a new view.
3. Select the location of the new view on the drawing.
4. Select an edge of, or axis through, the surface of the model in the view from which to create the auxiliary view. If the selected edge is from a view that has a trimetric or similar orientation, the system reorients the new view as it did the base feature section. Otherwise, it orients it with the selected surface parallel to the plane of the drawing.
   Note: After you create an auxiliary view, you can simplify or restore it without affecting its parent view.
5. Choose **Modify View** from the VIEWS menu, and then choose **Add Arrows** or **Del Arrows** from the VIEW MODIFY menu to show or delete arrows that point back to the view from which you created the projection.

Example: Auxiliary View
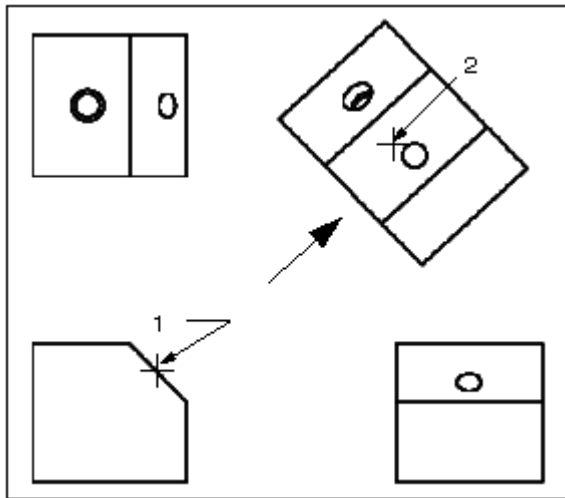
1   Select an edge of a surface that is perpendicular to the screen.
2   Placement pick.

Revolved Views

Revolved area cross-sectional views can be either full or partial views. Pro/ENGINEER creates a revolved area cross-sectional view from the parent view in which the cutting plane for the section is normal to the screen. It revolves the section 90 degrees around the cutting plane projection and offsets it along its length.

When you create this type of view, the system automatically creates a line of symmetry as well; however, this centerline is not an axis. You can move revolved views parallel to the cutting plane in the parent view, and their centering about the line of the cutting plane always remains the same. You can also modify the line of symmetry in various ways.

To Add a Revolved Cross-Sectional View

1. Choose **Views** > **Add View** > **Revolved** > **Section** > **Done**. The XSEC TYPE menu does not appear; the system creates the cross section automatically as an area cross section.
2. Select the location for the view, approximately along the cutting plane projection in the parent view; then select the parent view.
3. Choose the cross-section name from the menu. The revolved cross-sectional view appears in the default location, with all of the datum planes and axes temporarily shown. Pro/ENGINEER names it using the convention `revolved_<view_id>`.
4. Modify the centerline of symmetry, as necessary:
   − Replace the butting plane extension as the centerline of symmetry by selecting a datum plane or axis in the revolved view that is parallel to the cutting plane projection in the parent view. The system centers the revolved view so that the selected plane (or axis) is co-linear with the cutting plane in the parent view. If you press the middle mouse button instead, the cutting plane passes through the center of the revolved view.

   − Erase and resume the centerline by clicking **View > Modify View** and then clicking **Del Arrows** or **Add Arrows** in the VIEW MODIFY menu.

   − Clip the centerline by choosing **Edit > Move** on the menu bar. Select an endpoint of the centerline and the new location; then press the middle mouse button.

   − If you select a real axis (one that is normal to the screen) to center the view, display the axis by clicking **Axis** in the **Type** box of the **Show/Erase** dialog box (click **View > Show and Erase** on the menu bar).

Example: Revolved Cross-Sectional View

1   The cutting plane.

2   Revolved cross sectional view.

3   The parent view.
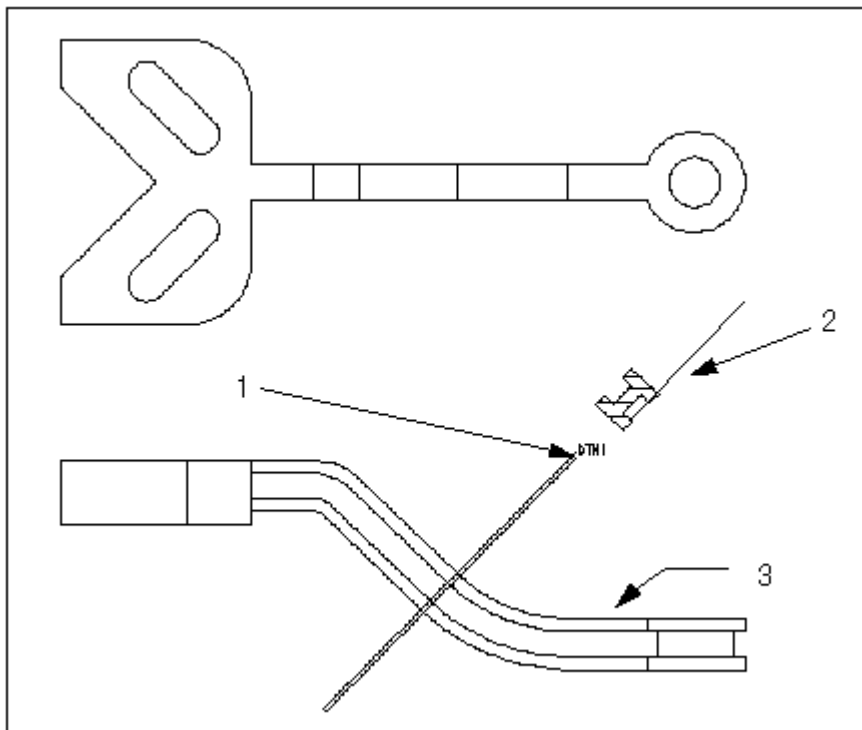
Graph Views

Using the **Graph** command in the VIEW TYPES menu, you can show views of graph features in a drawing. You would typically use a regular graph feature in a relation to control a variable section sweep for creating geometry. The graph is an actual datum feature that you would sketch in Sketcher mode using the **Datum** and **Graph** commands. The system shows the sketch of a graph feature and its dimensions in a drawing view and updates any changes parametrically. If there are no graph features for the current drawing model, the **Graph** command is not available in the VIEW TYPE menu.

A typical example of a part that uses a graph feature is a cam. A cam designer might want to show the cam profile to illustrate the linear motion of a cam follower as the cam rotates. The following figure shows a cam part and its corresponding profile represented by a graph feature that drives a variable section sweep. The system uses the variable section sweep to make the surface around the part; in other words, the graph feature (cam profile) controls the radius of the cam.
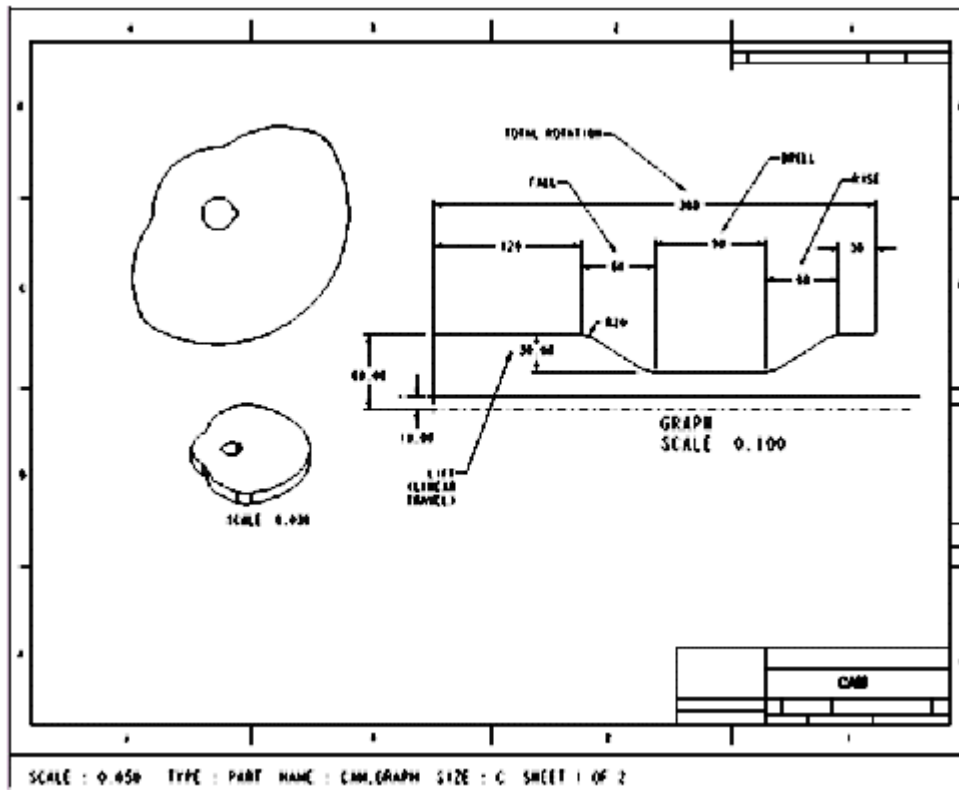
To Add a Graph View

1. Choose DRAWING > **Views** > **Add View** > **Graph**.
    Note: If there are no graph features for the current drawing model, the **Graph** command is not available in the VIEW TYPE menu.
2. Choose **No Scale** or **Scaled** and type a value. For scaled graph views, the scale label appears on the drawing below the view name.
3. Select a centerpoint for the drawing view.
4. From the GRAPHS menu, select a graph to show in the view. The system places the view, displaying the graph feature name below the bottom view boundary outline.
    Note: Since the graph feature is only two-dimensional, you do not need to orient it.

5. Make the following modifications, if necessary:
   – Change the view scale by selecting the value on the screen or choosing MODIFY VIEW > **Change Scale**.
   – Redefine the view type (from **No Scale** to **Scaled** or vice versa) by choosing VIEWS > **Modify View** > **View Type**.
   – Show the dimensions of the graph feature in the view by using the **Show/Erase** dialog box.
   – Show or erase the graph feature view name by using the **Show/Erase** dialog box.

Aligned Partial Views

Once you have a partial view in the drawing, you can create an aligned partial view by specifying the view boundary and indicating the direction of alignment relative to the existing partial view. You can also align an aligned partial view to a detailed view. You can align a partial view to another partial view in a drawing along the selected *straight* entity (such as an axis, edge, datum curve, or cable segment). This enables you to create multiple partial views that selectively show model geometry in the same view orientation, as well as maintain relative placement between these partial views.

The initial partial view—the *parent* view—determines various properties of successive aligned partial views, such as the following:

- The aligned view uses the same view attributes (for example, attributes specifying whether the view is scaled or exploded) as its parent view.
- You cannot change the scale of the aligned partial view independently, but it does change when you modify the scale of the parent partial view.
- You cannot reorient aligned partial views using the **Reorient** command directly (DRAWING **> Views > Modify View > Reorient**), since they are dependent views. However, if you reorient the parent view, the system realigns aligned partial views.

Because the aligned partial view is a dependent view, its location depends on its parent (the view to which it was aligned). Therefore, whenever you move the parent view, the location of the aligned view also adjusts. You can move the aligned view only in the direction of alignment, closer or farther away from the parent. If you erase the parent view or move it to another sheet, you can move the aligned view without restrictions until you resume the parent view or switch it back.

Note: An aligned partial view that you create using the **Copy & Align** command has its *own* local cross sections. That is, when you create it, it does not have the local cross sections of its parent view. You can add them and remove them later.

To Add an Aligned Partial View

1. Choose DRAWING **> Views > Add View > Copy & Align**, accept the defaults, and choose **Done**.
2. Select the center point for the view.
3. Select an existing partial view to which you are aligning the current view.
4. The aligned view temporarily appears in full. Select a reference point to define the center of the partial view.
5. Sketch a spline to define the boundary of the partial view.
6. On the current partial view, select a straight entity: edge, axis, datum curve, or cable segment. The current partial view aligns with the existing partial view along the selected entity.
7. Use **Ref Point**, **Boundary**, **Add Arrows**, **Del Arrows**, and **Snapshot** in the VIEW MODIFY (DRAWING **> Views > Modify View**)menu to modify the view as you would a regular projection view. You can modify the display of an aligned partial view for each partial view individually, regardless of the parent-child relationship that may exist between the views. If it does not have any dependent views, you can delete it. **Note:** The **Origin** and **Perspective** commands in the VIEW MODIFY menu, and the **Add Breakout** and **Del Breakout** commands in the VIEW BNDRY menu (**Views > Modify View > Boundary**) are *not* available for aligned partial views.

Example: Aligned Partial View
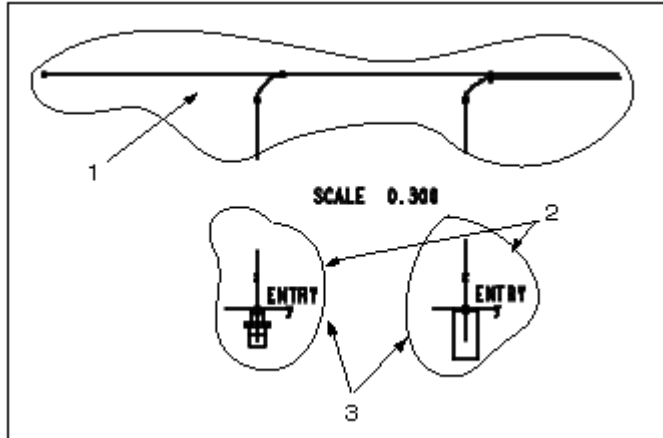
1   Parent view.
2   Aligned partial view.
3   Use these segments to establish the direction of alignment.

About Modifying Views

Using commands in the VIEW MODIFY menu, you can perform the following procedures:

- Rename a view
- Change the scale of a view
- Redefine views by changing the view type
- Change the process state of a process assembly view
- Reset the view origin
- Change the view alignment
- Change or resketch view boundaries
- Change the explosion distances of exploded views
- Exclude all graphics behind a specified plane using Z-Clipping
- Modify, remove, or replace view cross sections
- Change crosshatch

In addition, using the **Disp Mode** command in the VIEWS menu, you can modify the display mode in various ways.

To Rename a View

Using the **View Name** command in the VIEW MODIFY menu, you can change the name of a view in session.

1.   Choose VIEWS **> Modify View > View Name**.
2.   Select the view to rename.
3.   Enter the new name.

View Type Change Restrictions

Using the **View Type** command in the VIEW MODIFY menu, you can redefine drawing views in the following ways:

- Change the orientation type among these view types:
    – Projected, auxiliary, and general

- Full, half, and partial
- Change the orientation type between these view types:
  - Total cross section and area cross section
  - Scale and no scale
  - Exploded and unexploded

When redefining a view, the following restrictions apply:

- You cannot redefine views that are defined as **Of Flat Ply**, or **Copy & Align**.
- You cannot redefine a projection view to a broken view.
- You cannot change the following view types to a view of the same type, showing a different portion of the model:
  - Broken view
  - Perspective view
  - Of Surface view
  - Revolved view (the only possible change for a revolved view is from partial to full, or from exploded to unexploded)
  - View with a total unfolded cross section
  - View with an unfolded area cross section
  - Aligned view

Note: You cannot modify the view type of the first, or general, view.

To Change the View Type

Note: You cannot modify the view type of the first, or general, view.

1. Choose VIEW MODIFY > **View Type**.
2. Select a drawing view to redefine.
3. Choose commands from the VIEW TYPE menu. (If the view is to be cross-sectional, the XSEC TYPE menu appears with all current commands highlighted. You can then change between commands in the top half of the menu, and among **Area Xsec**, **Total Align**, and **Total Xsec**.)
4. After you make all modifications to the view, the system redraws the view. If you modify the view to projected or auxiliary, and it was a general view previously, the system highlights all views that the change could affect, and you must specify if you still want to reorient the view. If you choose not to reorient it, it remains general.

   When defining or redefining a projected or auxiliary view, keep in mind the following:
   - For a projected view, you must select its parent view. For an auxiliary view, you must select a face or axis as a reference, as in view creation.
   - When you convert a projected or auxiliary view to general, you do not have to reorient the view. Instead, the system copies the orientation information from the parent view.

About Changing the View State

Using the **View State** command in the VIEW MODIFY menu, you can modify views of assembly drawings, as well as Pro/PROCESS for ASSEMBLIES or Pro/PROCESS for MANUFACTURING drawing views, in the following ways:

- Change the process state
- Modify the simplified representation
- Replace the simplified representation

You can also use the **View State** command to change the simplified representation of the model of a drawing view.

Note: The **View State** command is available only if you are working with a process state view or models of simplified representations.

To Change the View State

You can use this procedure to change the view state, or to change the state of the model on an existing view.
1. Click DRAWING > **Views** > **Modify View** > **View State**.
2. Select the view whose state you want to change. The **Process State** dialog box or the **Open Rep** dialog box opens containing a list of the available modification options.
3. Set the desired options for changing the view state or the model state on an existing view. Click **OK** when you are done.

Resetting the Origin

By default, the origin of a drawing view is in the center of its outline. Using the **Origin** command in the VIEW MODIFY menu (DRAWING > **Views** > **Modify View** > **Origin**), you can reset the origin of a drawing view by parametrically referencing model geometry. This attaches the selected geometry to its current location on the drawing and prevents it from shifting whenever the model geometry changes.

You can reset the origin of general, auxiliary, and broken views any time after you create them, but it does not change the current position of the view. The effect of the new origin is noticeable only when the system updates the views to reflect changes in model geometry. However, if you reset the origin of a perspective view, it *alters* the current position of the view because the view origin is part of the orientation.

Note: You cannot change the view origins of total unfolded cross-sectional views.
To select the point for the origin, you can choose a model edge, datum curve, datum point, coordinate system, or cosmetic feature entity. However, when selecting a point, keep in mind the following:

- For a general view, the selected point becomes fixed.
- For a projection and auxiliary view, the system passes the selected point onto a ray passing through the origin of the parent view in the direction of projection. This projected point becomes the origin of the view.
- To select a coordinate system as the origin of a newly created view or of a modified view, specify its name as the value for the configuration file option "drawing_view_origin_csys." If you do not want the system to use a coordinate system that you set previously, specify the value as "no."

To Reset the Origin of a View

1. On the DRAWING menu, choose **Views > Modify View > Origin** > **On Item**.
2. Select a view.
3. Select a point to indicate the origin of the view. Use commands in the GET SELECT menu, if necessary.
   Note: If the geometry referenced in a view is suppressed or deleted, the system warns you that model geometry is missing. For views oriented with the suppressed or deleted reference, the view returns to its default orientation.
4. After setting the origin of the view, you can proceed to set the origin of another view, or choose **Done/Return** on the VIEWS menu.

Tip: Resetting the Origin of Broken Views

You can set the origin of a broken view at any time in any one of the members using the same method that you would use to set the origin of a general, projection, or auxiliary view. The new set origin is the origin for all of the members. You can unset it at any time by choosing **At Center** from the VIEW ORIGIN menu (DRAWING > **Views** > **Modify View** > **Origin** > **At Center**). If a broken view does not have an origin, the default view center is at the center of the 3-D model in the first broken view.

To Align a View

1. Click DRAWING > **Views > Modify View > Alignment > Align View**.
2. Select the view that you want to align.
3. Select the view to which you want to align the highlighted view. You can align the view either vertically or horizontally.
4. After you select the view to align to, the first selected view snaps to its new location and is now aligned with the second view.
   Note: The view will remain aligned and move like a projection view until it is unaligned.

To Unalign a View

1. Click DRAWING > **Views > Modify View > Alignment > Unalign View**.
2. Select the view to unalign.
3. Click **Done Sel** or the middle mouse button. The view is no longer aligned with any other view.

To Modify the Alignment of a View

1. Click DRAWING > **Views > Modify View > Alignment > Modify Align**.
2. Select a view to modify. Click **Confirm**.
3. Click **Parent** or **Child** to determine an alignment base point on the parent or child.
   – Click **Origin** to make the alignment base point the view origin.
   – Click **Selection** to select a point in the view to be the alignment base point.
4. The child view snaps to its new location, using the new alignment base point.
   Note: When the selected view is a parent/child to other views, all of the related views highlight in blue and may also move when you modify the alignment.

Changing the Boundaries

You can modify the outer borders of detailed, partial, and broken views by moving their reference points. You can also resketch the view boundaries.

To Modify a Reference Point

1. Choose DRAWING **> Views > Modify View > Ref Point**.
2. Select a view to modify.
3. The system highlights the reference point in red. Select a new location for it:
   – For a *detailed* view, select the parent view.
   – For a *partial* view, select the view inside the breakout boundary. If you want to choose a point outside the view boundary, modify the breakout first.
   – For a *broken* view, reselect the reference point for each side of the view.

To Resketch a Boundary of a Detailed View

1. Choose DRAWING **> Views > Modify View** > **Boundary**. The VIEW BNDRY menu opens.
2. Select a detailed view.
3. Choose VIEW BNDRY > **Mod Breakout**.

To Erase or Show the Outer Boundary

1. Choose DRAWING **> Views > Modify View > Boundary**.
2. Select a detailed or partial view.
3. Choose **Erase Outer** or **Show Outer**; then choose
   VIEW BNDRY > **Done**. The system updates the view.
   Note: You cannot display or erase the border of a detailed or partial local cross-sectional view.

About Excluding Graphics Behind a Specified Plane

If you have a license for Pro/DETAIL, you can specify a plane parallel to the screen and exclude all graphics behind it by performing Z-Clipping.

When you perform Z-Clipping in a view, keep in mind the following:

- If the system cannot regenerate the reference for the clipping plane, Z-Clipping does not take effect for the view (an error message appears).
- The Z-Clipping of a detailed view is always the same as that of its parent. You cannot modify it individually.

To Exclude Model Graphics Behind a Specified Plane

1. Choose DRAWING **> Views > Modify View > Z-Clipping**.
2. Select a view to modify. The Z-CLIPPING menu opens.
   Note: You cannot perform Z-Clipping in the following types of views: unfolded cross-sectional, area cross-sectional, exploded, and perspective.
3. Choose Z-CLIPPING > **Add/Change**.
4. For the reference point, select an edge, surface, or datum plane that is parallel to the view. You can place it on any view. The reference point appears in magenta in the selected view and in any of its projected or auxiliary views (if the selected view is a projected or auxiliary view, the reference point appears in magenta in its general view).
   All geometry behind the defined Z-Clipping does not appear, but any geometry that the plane contains entirely does appear. The system clips geometry that intersects the plane. Only the portion in front of the plane appears.

Example: Excluding Graphics Behind a Specified Plane
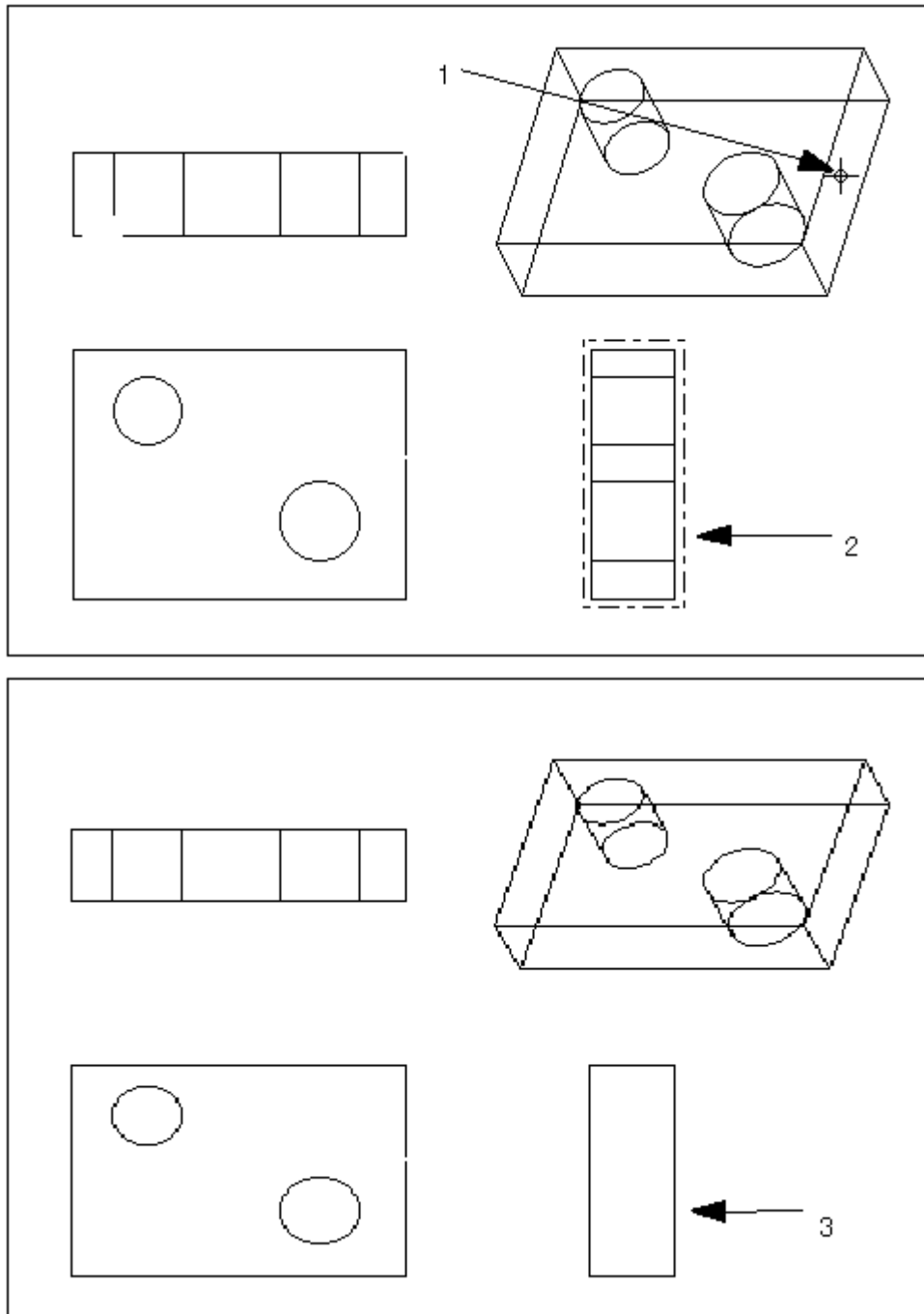
1   Select this surface as the reference point.
2   Select this view.
3   The geometry intersecting the plane is clipped.

About Specifying the View Scale

As you create a view, you can specify an independent scale or a *predefined* scale value. To create the view with an independent scale value shown under the view, choose **Scale** from the VIEW TYPE menu (DRAWING > **Views > Scale**). To create a scaled view with the default scale shown in the lower-left corner of the screen, use the **No Scale** command. By setting the configuration file option default_draw_scale, you can control the initial setting of the default scale; otherwise, the system bases it on the size of the model.

Modifying the Scale

A newly created view takes on the value of the drawing scale, unless you specify a different scale value. You can change the scale of any existing view. A modified view scale changes independently of the drawing scale. If you change the drawing scale, only those scales for views whose scales have not been modified and are therefore equal to the drawing view remain dependent on the drawing scale. To show or erase the view scale note, use the **Show/Erase** dialog box (click **View > Show and Erase** on the menu bar).

Drawing Scale

The drawing scale value, displayed in the lower-left corner of the drawing sheet, represents the drawing-to-model scaling. For example, for a value of 0.25, Pro/ENGINEER scales the drawing views at one-quarter (1/4) of the actual model size.

When creating a new view on a drawing sheet, if you do not enter a value for the view scale, the scale takes on the value of the drawing scale. You can *scale a view* (change the scale of an existing view). The drawing view scale does not change with changes in individual view scale. In other words, the drawing scale parameter shown at the bottom left corner of the main window controls the scale for all views except detailed and scaled views.

To Modify the Drawing Scale

1. On the menu bar, click **Edit > Value**.
2. At the lower-left corner of the drawing window, select the label or value for the Scale (drawing scale).
3. At the prompt, enter a new drawing scale value.

All views of the active drawing model (except for detailed and scaled views) change in size corresponding to the new drawing scale value.

View Scale

Using the **Change Scale** command in the VIEW MODIFY menu (DRAWING > **Views > Modify View > View Scale**), you can modify the scale of a view. However, if the parent view of a projection view is a scaled view, you cannot use this command to modify the scale of the projection view.

To Change the Scale of a View

You can change the scale value for a scaled view using object-action or action-object orientation.

Method 1

1. Select a scaled view.
2. Press the right mouse button and select **Change Scale** in the shortcut menu.
3. Type the new scale value. The system modifies the view scale and redisplays the view.

Method 2

1. Click DRAWING > **Views > Modify View > Change Scale**.
2. Select the view whose scale you want to modify, and then type the new scale value.

About Modifying Cross Sections

Using commands in the VIEW MODIFY menu (DRAWING > **Views** > **Modify View**), you can modify cross sections in the following ways:

- Change the reference points of a local cross section in a general or projection view
- Change the boundary of a local cross section in a general or projection view
- Create a full cross-sectional view when deleting a breakout
- Modify a partial view with a local cross section
- Modify a broken view with a local cross section
- Remove or replace a cross section
- Flip a full total cross-sectional view

Using additional commands, you can also display set datum planes in area cross-sectional views and move cross-section arrows or cross-section text. To redefine the side and direction of offset cross sections, you can use the XSEC MODIFY menu.

To Modify the Reference Points of a Local Cross Section

1. Choose DRAWING > **Views** > **Modify View** > **Ref Point**.
2. Select a general or projection view. The system highlights boundaries and reference points of the local cross sections.
3. Select a reference point and move it.
4. You can continue to modify other local areas. When you have finished, choose EXIT > **Done**.

To Modify Boundaries of a Local Cross Section

1. Choose DRAWING > **Views** > **Modify View** > **Boundary**.
2. Select a general or projection view. The system highlights boundaries and reference points of the local cross sections.
3. Choose **Breakout** from the BROKEN VIEW menu, and then choose **Mod Breakout** from the VIEW BNDARY menu.
4. Select a boundary and resketch it.
5. When you have finished, choose VIEW BNDRY > **Done**.

To Create a Full Cross-Sectional View When Deleting a Breakout

1. Choose DRAWING > **Views** > **Modify View** > **Boundary** > **Del Breakout**.
2. Select a breakout. If it is the last breakout, specify whether you want it to be a full cross-sectional view.
3. If you type [yes], the system creates a full cross-sectional view. When you have finished, choose VIEW BNDRY > **Done**.

To Modify a Partial View with Local Cross Sections

1. Choose DRAWING > **Views** > **Modify View** > **Boundary**.
2. Select a partial view.
3. Using commands in the VIEW BNDRY menu, do one or all of the following:
   – Resketch a spline boundary of a local cross section by choosing **Mod Breakout**.
   – Relocate a reference point for a local cross section by choosing **Move Ref Pnt**.
   – Add a new local cross section by choosing **Add Breakout**.
   – Delete a local cross section by choosing **Del Breakout**.

4. When you have finished, choose VIEW BNDRY > **Done**.

To Modify Reference Points of a Broken View

1. Choose DRAWING **> Views > Modify View** > **Ref Point**, and select a broken view.
2. Do one of the following:
   – To modify the reference points of local cross sections, *go to Step 5*.
   – To modify the reference points of outer boundaries of the broken view, choose MOD REF POINT > **View Break**. The system displays the current reference points.
3. Select a reference point that you want to redefine and specify a new location. Do the same for any other reference points that you want to modify.
4. After you have specified the new locations, choose GET SELECT > **Done** and EXIT > **Done**. The system updates the view.
5. To modify the reference points of local cross sections on the broken view, choose MOD REF POINT > **Breakout**.
   Note: Half views cannot have breakouts. If you choose **Half** and the view has breakouts, the system removes them if you choose **Done**.
6. Type [Y] to temporarily display the broken view in full. The current reference points appear.
7. Select a reference point and select its new location.
8. After modifying the view, choose EXIT > **Done**.

To Modify a Boundary or Local Cross Sections of a Broken View

1. Choose DRAWING **> Views > Modify View > Boundary**.
2. Select a view to modify.
3. To modify local cross sections, *go to Step 6*. To modify the outer boundaries, choose BROKEN VIEW > **View Break**.
4. Select lines that share the same splines; then choose **Done Sel.**
5. Resketch the spline; then choose EXIT > **Done**.
6. To modify local cross sections on the broken view, choose BROKEN VIEW > **Breakout**.
7. Type [Y] to temporarily display the broken view in full.
8. Choose VIEW BNDRY > **Mod Breakout** and select a boundary. Resketch it; then choose **Done**.

To Remove or Replace a View Cross Section

1. Choose DRAWING **> Views > Modify View > X-Section**.
2. Select the drawing view that contains the cross section to remove or replace.
3. Select a cross-sectional portion of the view, as you would to modify crosshatch (if the view is a cross-sectional view with breakouts, selecting in a breakout selects its cross sections; selecting in the view, but outside any breakout, selects the outer cross section).
4. Choose a command from the XSEC ENTER menu:
   – If you choose **Create** to make a new cross section, you must select a view in which to display arrows if they are not already shown. If the same cross section reappears in multiple breakouts, the new cross section replaces the old one in all of its breakouts. (You cannot reassign only some of the breakouts from one cross section to another. To do so, use **Boundary** and delete or recreate the breakouts.)
   – If you choose **Retrieve** and then retrieve the same cross section, but it is a planar cross section, the system resets the viewing direction of the cross section.
   – If you choose **Retrieve** and then retrieve an offset cross section, choose **Flip** or **Okay**.
   – If you choose **None**, the system strips the selected cross section and all of its breakouts from the view and its detailed children.

   The system updates the view, and all details reappear.
   Note: When the system regenerates a cross section in a drawing, and you add or remove model geometry, it creates new edges in the drawing that you can reference. Because these edges are permanent additions to the

system, the model changes when those edges are first created. The *model* owns the cross section.

Flipping a Full Total Cross Section

Using the **Flip** command in the MOD VW XSEC menu (DRAWING **> Views > Modify View > X-Section**), you can flip a full total cross-sectional view to retain the opposite half of the cut part. This command is modal—if you want to flip another full total cross section, you can immediately select another view. To replace the cross section with another, choose **Replace** from the MOD VW XSEC menu. The **Flip** command switches—between one side of the model and the other—the portion of the model removed by the cross-sectional cut. The cross-section arrows that appear on a perpendicular view flip their orientation to reflect this change. They flip because cross-section arrows always point toward the portion of the model that remains after using the cross section to remove material from a model view.

Note: Using the **Flip** command does not reorient the model in the view. It flips only the cross section. You should choose **Reorient** from the VIEW MODIFY menu to create a new view orientation for the cross-sectional view.

To Delete a Cross Section from a Model

Using the **Delete** command in the MOD VW XSEC menu, you can delete a cross section from a model without having to retrieve the model.

1. Choose DRAWING **> Views > Modify View > X-Section > Delete**.
2. From the DRAW MODELS menu select the name of the model from which to delete a cross section.
3. From the XSEC NAMES menu select the name of the cross section to delete. The cross section is removed from the model.
   Notes:
   – You cannot delete a cross section when it is being used by a drawing in session.
   – Deleting a cross section in the drawing also deletes it in other drawings.

To Rename a Cross Section in Drawing Mode

Using the **Rename** command in the MOD VW XSEC menu, you can rename a cross section currently in use in a drawing without having to enter Part or Assembly mode to retrieve a model.

1. Choose DRAWING **> Views > Modify View > X-Section > Rename**.
2. Select a view that contains the cross section to rename and enter the new name.
   The system updates the name of the cross section in the current drawing, in the model that contains the cross section, and in any drawing that already was using the cross section.

To Display Cross Section Datum Planes

Using the **Show/Erase** dialog box (**View > Show and Erase** on the menu bar), you can display set datum planes in area cross-sectional views. Set datums that you display this way behave identically to those in other views; you can perform all usual detail actions on them, including dimensioning.

Moving Cross-Section Arrows and Text

If you have a Pro/DETAIL license, you can move cross-section arrows or cross-section text. When you select a cross-section arrow, you can move it independently of the other cross-section arrowhead. To move the arrows simultaneously, select the arrow line. To change the cross-section text, change the name of the cross section in either Part or Assembly mode.

For more information about moving items in Drawing mode, refer to the topic To Move or Clip Items in Drawing Mode.

To Move Cross-Section Arrows or Text

1. Do one of the following:
   - Select text to move independently of the arrows
   - Select an arrowhead to move the arrow and text together, independently of the other cross-section arrowhead
   - Select an arrow line to move the arrows simultaneously
2. Click to place the arrow in the new location.

To Modify Cross-Section Text

1. On the menu bar, click **Format > Text Style**.
2. Select any of the cross section names.
3. In the **Text Style** dialog box, clear the **Default** check box for **Height**, and type a new value.
4. Clear the **Use Default** check for **Width Factor**, and type a new value.
5. Click **Apply**. The height and width of the selected item changes.
6. To reset the text to the old style, click **Reset Settings**, and then click **Apply**.
   Note: To show or erase a cross-sectional view name, use the **Show/Erase** dialog box (**View > Show and Erase**).

To Control the Cutting Line Display

You can control the display of the cutting line in a cross-sectional view by setting the following drawing setup file options.

1. Click DRAWING > **Advanced > Draw Setup**. The **Options** dialog box opens.
2. Set the following drawing setup file options:
   - `cutting_line`—Determines the standard (ANSI, ISO) for the cutting line display
   - `cutting_line_adapt`—Sets the display of all line fonts used to display cross-sectional views adaptive so that they begin in the middle of a complete line segment and end in the middle of a complete line segment
   - `cutting_line_segment`—Controls the length of the thickened portion of the cutting line

To Redefine Offset Cross Sections

You can redefine the side and direction of offset cross sections by using the **Redefine** command in the XSEC MODIFY menu in either Part or Assembly mode, *not* Drawing mode.

1. In Part or Assembly mode, choose **X-Section** and **Modify Xsec**.
2. Select the name of the cross section from the XSEC NAMES menu.
3. Choose XSEC MODIFY > **Redefine** and one of these commands from the REDEFINE menu:
   - **Attributes**—Redefines the cross section from one side or both sides
   - **Direction**—Redefines the direction
   - **Section**—Redefines the cross section of the offset cross section
   - **Scheme**—Redefines the scheme of the offset cross section

About Working with Draft Cross Sections

Using **Edit > Fill** and then **Hatched** or **Solid** in the menu bar, you can fill or hatch a draft cross section. Pro/ENGINEER defines a draft cross section boundary by a closed loop of draft geometry.

When working with draft cross sections, keep in mind the following:

- You can define islands, or unfilled and uncrosshatched areas, within the closed loop.
- Islands cannot intersect the outer boundary, nor can they intersect each other.
- You can include any number of loops.

- You can include both types (filled and crosshatched) in nonparametric drawing formats, if desired.
- The system does not consider a draft cross section to be a single entity. Therefore, to place it on a layer, you must add the contour and crosshatch pattern individually.
- You can modify only one draft cross section at a time. You cannot modify a draft cross section that contains open loops.
- You can delete either the entire cross section (with the cross-hatching or filling and boundary) or the cross-hatching/filling.

Example: A Draft Cross Section
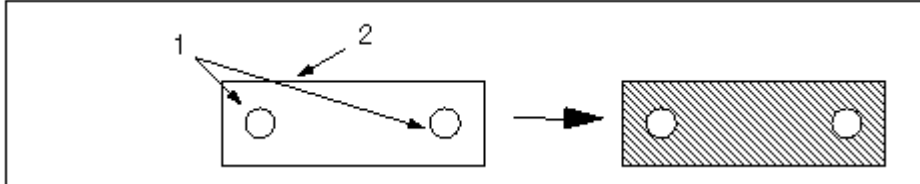
1   Define islands.
2   Define outer boundaries.

To Create a Draft Cross Section or Filled Area

1. Select a closed draft entity, or a series of entities that form a closed area, from which you want to add a draft cross section. **Note:** You can use the **Pick Many** command to select multiple entities. Click *See Also* below for more information.
2. On the menu bar, click **Edit > Fill**, and then select **Hatched** or **Solid**.
3. At the prompt, enter a name for the cross section. When you press ENTER, the draft cross section is created.

Using Pick Many to Select Entities

You can use **Pick Many** to select entities in a closed loop. The system highlights incorrect entities (that is, those that do not form a loop) in magenta. If the entities almost form a loop, but leave a slight gap, it highlights the endpoints of the entities at the gap.

To Modify a Draft Cross Section

1. Select the draft cross section by selecting the cross section boundary or on the cross-hatching.
2. Click **Edit > Properties**. The MOD XHATCH menu opens in the Menu Manager.
3. Choose commands from the MOD XHATCH menu. You can retrieve a previously saved crosshatch pattern to use in the draft cross section.

To Delete a Crosshatched or Filled Area Without Deleting the Boundary

1. Select the crosshatched area you want to delete, and then click the right mouse button to display the shortcut menu.
2. Choose **Delete** from the shortcut menu.
3. Select inside the crosshatched or filled area.
4. Choose DWG SELECT > **Done Sel**. The system deletes the cross-hatching or filling, leaving the bounding entities. If the border is not visible, choose **Repaint**.
   Note: To delete the entire cross section *with its boundary*, select a point on the contour; then choose **Done Sel**.

About Grouping Draft Entities

Using the **Group** command in the TOOLS menu, you can create, suppress, resume, and explode draft groups. Pro/ENGINEER acts upon all of the entities within a group in the same way. However, modifying the line style of a grouped entity affects *only* that entity.

When grouping entities, keep in mind the following:

- You cannot use other commands such as **Intersect** or **Trim** to modify entities belonging to a group.
- You can put draft entities on a layer and blank them all at once by selecting that layer. You can also blank individual draft entities without having to blank all entities on a layer.
- You can include notes, symbols, and geometric tolerances, except those created using **On Item**, in draft groups.
- The system creates a draft cross section as a group containing the cross-hatching and the bounding entities. You cannot explode the cross-hatching, but you can delete it.

To modify grouped notes, dimensions, and symbols, use **Format > Text Style** in the menu bar as you would for notes.

To Create a Draft Group

1. Choose DRAWING **> Tools > Group** > **Create**.
2. Using the GET SELECT menu, select all of the entities to include in the group.
3. Type a group name.

To Suppress a Draft Group

1. Choose DRAWING **> Tools > Group**, and then on the DRAFT GROUP menu, choose **Suppress**.
2. Using the GROUP ACCESS menu, do one of the following:
   - Choose **Select** and select a group.
   - Choose **By Name** and type the group name.

To Resume a Suppressed Group

1. Choose DRAWING **> Tools > Group**, and then on the DRAFT GROUP menu, choose **Resume**.
2. Do one of the following:
   - Select the group to resume from the GROUP NAMES menu.
   - Choose **Names** and type the group name.

To Explode a Resumed Group

1. Choose DRAWING **> Tools > Group**, and then on the DRAFT GROUP menu, choose **Explode**.
2. Do one of the following:
   - Select the group to explode by selecting one of its entities with the mouse.
   - Choose GROUP ACCESS > **By Name** and type the group name.

To Modify a Draft Group

1. Choose Choose DRAWING **> Tools > Group**, and then on the DRAFT GROUP menu, choose **Edit**.
2. Do one or both of the following, as necessary: Do one of the following:
   - Select a group by selecting one of its entities with the mouse.
   - Choose GROUP ACCESS > **By Name** and type the group name.
   - Add draft entities to the group by choosing **Add** and selecting the entities to add to the group.

   Note: You cannot add draft datums, 3-D datums, draft axes, symmetry lines, or balloons to draft groups.
3. Delete entities from the group by choosing **Remove** and selecting the entities to remove from the group.

About Cut, Copy and Paste

You can manipulate detail items using cut, copy, and paste functionality. When you cut, copy or paste items such as notes, symbols, draft entities and tables, the selected items are copied temporarily to the clipboard and then pasted onto the same sheet, a different sheet, or a different drawing.
The detail items that you can cut, copy and paste are as follows:

- Notes (with and without leaders)
- Balloons
- Symbols
- Draft entities
- Draft dimensions
- Tables

Notes, balloons, and symbols with leaders are copied with their leader. The leader is copied as it looks on the source drawing. You are able to move the end of the leader on the target drawing by dragging the endpoint and jog to the desired location.

The parameters in the notes are copied as parameters and are evaluated on the target drawing. If the parameter cannot be evaluated on the target drawing, three asterisks are displayed to indicate this.

Tables with regular text (no repeat regions) are copied as text. When pasted, the text looks as it does on the source drawing. Tables with repeat regions are copied with the report symbol names and have to be updated on the target drawing using **Update Tables**.

The **Cut**, **Copy**, and **Paste** commands are available from the **Edit** menu or directly from the toolbar icons on the top toolbar. They allow you to copy detail items and paste them between drawings, reports, formats, layouts and diagrams. The following from - to combinations are supported:

- Drawing – Drawing, Report or Format
- Report – Drawing, Report or Format
- Format – Drawing, Report or Format
- Layout – Layout (limited to same type only)
- Diagram – Diagram (limited to same type only)

Keyboard accelerators can also be used to perform cut, copy and paste functions:

- Cut—CTRL+X
- Copy—CTRL+C
- Paste—CTRL+V

The **Cut** command works similarly to the **Copy** command in that the selected detail item is copied to the clipboard. However, the **Cut** command moves the selected item from its original location to its new location.

To Copy Detail Items on the Same Sheet Using the Clipboard

1. Click **Edit > Copy**, or click on the toolbar.
2. Select the detail items you want to copy to the clipboard, and then click . The **Drawing Clipboard** dialog box opens and contains the copied detail items.
3. Define the translation origin point by picking a point in the **Drawing Clipboard** dialog box. The translation origin point appears as a yellow square.
4. Select the location on the drawing where you want to place the copied items. This action pastes the copied items.

Note: If you want to move detail items, use **Cut** instead of **Copy** .

To Copy Detail Items from one Sheet to Another Sheet Using the Clipboard

1. Click **Edit > Copy**, or click ⬚ on the toolbar.
2. Select the detail items on the source sheet you want to copy to the clipboard.
3. Move to the target sheet by working from the **Sheets** menu or by using the **Sheets** spin box on the toolbar.
4. When you are in the target sheet, click ⬚. The **Drawing Clipboard** opens and contains the copied detail items from the source sheet.
5. Define the translation origin point by picking a point in the **Drawing Clipboard** dialog box. The translation origin point appears as a yellow square.
6. Select the location in the target sheet where you want to place the copied items from the source sheet. This action pastes the copied items.

Note: If you want to move detail items, use **Cut** ⬚ instead of **Copy** ⬚.

To Copy Detail Items from one Drawing to Another Drawing Using the Clipboard

1. Click **Edit > Copy** on the menu bar, or click ⬚ on the toolbar.
2. Select the detail items on the source drawing you want to copy to the clipboard.
3. Select the target drawing from the **Window** menu on the menu bar. This action opens the target drawing.
4. Click **Edit > Paste** on the menu bar, or click ⬚. The **Drawing Clipboard** opens containing the copied detail items from the source drawing.
5. Define the translation origin point by picking a point in the **Drawing Clipboard** dialog box. The translation origin point appears as a yellow square.
6. Select the location in the target drawing where you want to place the copied items from the source drawing. This action pastes the copied items.

Note: If you want to move detail items, use **Cut** ⬚ instead of **Copy** ⬚.

To Copy Detail Items from a Drawing to Another Drawing

Using the **Copy** command from the TOOLS menu, you can copy detail items, such as drawing tables, notes, symbols, draft entities, and balloons, from one drawing to another. Items commonly used on all drawings can be stored in one main drawing to be copied to new drawings.

1. Choose **Edit > Copy from Other Drawing**.
2. In the **Open** dialog box, select the name of the source drawing from which to copy items. The selected drawing is displayed in another Pro/ENGINEER window, and the DETAIL ITEMS menu appears.
3. If the source drawing contains more than one sheet, use the commands in the PLACE SHEET menu to switch to the correct sheet, and choose **Done**.
4. Choose a command from the DETAIL ITEMS menu to specify the items to copy. If you choose **Draft Items** or **Dwg Tables**, select the items to copy and choose **Done Sel**; then click **Done**.
   - **Entire Sheet**—Copies all the current items on the current sheet onto the other drawing.
   - **Draft Items** (selected by default)—Allows you to select draft items to copy from the source drawing.
   - **Dwg Tables**—Allows you to select drawing tables to copy onto the other drawing.
   - **Done**—Copies selected items to the target drawing.
   - **Quit**—Cancels current selections and returns to the TOOLS menu.
5. Select a point on the source drawing to define the first point of the translation vector. This selected point indicates where the items will be placed on the target drawing.

6. Select a point on the target drawing to define the final point of the translation vector. The items are placed on the drawing relative to the first point of the translation vector.
After the selected items are copied to the target drawing, the system returns to the TOOLS menu.
You do not have to scale the items after they have been placed on the target drawing. The system automatically scales the items copied between drawings of different size. For example, when items are copied from an E size to a B size drawing, the items are scaled by the same factor.

Rules for Copying Detail Items from Drawing to Drawing

The following rules and restrictions apply to copying detail items from drawing to drawing:

- The source and target drawings must use the same units.
- Notes, balloons, and symbols with leaders are copied with their leader. The leader is copied as it looks on the source drawing. You can move the end of the leader on the target drawing by dragging its endpoint and jog to the desired locations.
- Notes that contain dimension parameters (that is, &d23) cannot be copied. Notes that contain other parameters can be copied.
- Tables are copied as follows:
  - Tables without repeat regions are copied as text and look the same as they do on the source drawing.
  - Tables with repeat regions are copied with the report symbol names and have to be updated on the target drawing using **Update Tables**.

To Translate and Copy Draft Entities

Using the **Copy** command in the TOOLS menu, you can copy draft entities or free notes by translating them in an x- and/or y-axis direction, or by rotating them about a point. To translate or rotate the draft entities without creating any copies, use the commands available in the MOD ENTITY menu.

Note: You *cannot* rotate symbols.
1. Choose TOOLS > **Copy** > **Translate**.
2. Select all of the entities to copy and translate. Choose **Done Sel**.
3. Choose **Horiz**, **Vert**, or **Ang Length** from the GET VECTOR menu to specify the direction for translating the new entities.
4. Type the number of copies to create and press ENTER. The system copies the entities and moves them

The GET VECTOR Menu

You can use the GET VECTOR menu for various operations, such as translating and copying draft entities. In this case, you use the GET VECTOR menu to specify the direction for translating new draft entities.

- **Horiz**—Translates the entities along the horizontal direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the right of the sheet.
- **Vert**—Translates the entities along the vertical direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the top of the drawing sheet.
- **Ang/Length**—Translates the entities at an angle, and a specified distance in that direction, measuring the angle relative to the horizontal in the counterclockwise direction. An arrow appears showing the positive direction of translation.
- **From-To**—Translates the entities along a vector defined by using a start point and endpoint.

To Rotate and Copy Draft Entities

1. Choose TOOLS > **Copy** > **Rotate**.
2. Select all of the entities to copy and translate. Choose **Done Sel**.
3. Type a rotation angle for the entities and press ENTER. The system measures the angle from the horizontal

in the counterclockwise direction.
4. Select the center point for the copy rotation using GET POINT menu commands.
5. Type the number of copies to create and press ENTER. The system creates the copies and rotates them.

To Mirror an Entity

Using the **Mirror** command in the TOOLS menu, you can create copies of draft entities, unattached symbols, and unattached notes by mirroring them about a draft line.

1. Choose TOOLS > **Tools** > **Mirror**.
2. Select detail items to mirror; then choose **Done Sel**.
3. Select a draft line about which to mirror the entities. The system creates a copy of the selected entities as mirror image of the source entities.

Example: Mirroring Entities
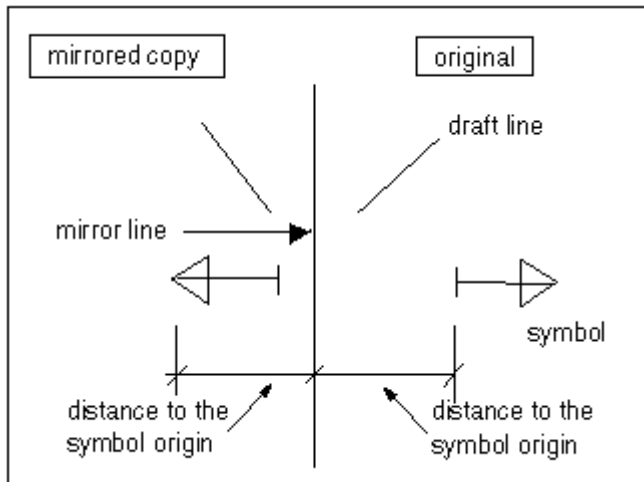
To Break Two Entities at their Intersection

1. Choose DRAWING **> Tools > Intersect**.
2. Select two intersecting entities.
   The system creates breaks at the intersection of both entities, thereby creating four separate entities. If there is more than one possible intersection, such as a spline crossing a line in several places, the system creates the intersection closest to the point you selected. Intersected construction geometry becomes regular draft geometry.

Note: If the selected entities do not actually intersect, the system does not find an intersection. In this case, use the **Trim** command.

To Trim Draft Geometry

Using the **Trim** command in the TOOLS menu, you can lengthen or shorten draft geometry. The system uses the geometry definition to find its intersection with the bounding entity.

1. Choose DRAWING **> Tools > Trim**.
2. Choose a command from the TRIM menu.
3. Select the entity to trim (you *cannot* trim construction geometry). To shorten the entity, select the portion of the entity that you want to keep in the drawing. To lengthen it, select it anywhere.
4. Choose **Done Sel**.

Tip: When a Spline Does Not Intersect the Bounding Entity

When a spline or the continuation of an arc does not intersect the bounding entity, the system cannot calculate the intersection, so it does not trim the entity.

The TRIM Menu

The following are the options on the TRIM menu, accessed when you choose DRAWING **> Tools > Trim**.

- **Bound**—Trims to a specified entity or point.
- **Length**—Trims to a specified length.
- **Increm**—Trims or extends by a specified amount.
- **Corner**—Trims two entities to their intersection.

To Offset Draft Geometry

Using the **Offset** command in the TOOLS menu (DRAWING **> Tools > Offset**), you can offset draft geometry from other draft entities and model geometry. You can select an axis line, datum plane, draft datum plane, or draft datum axis for creating an offset draft entity. This axis line can come from any shown axis, without regard to orientation. When you create an offset from the axis perpendicular to the screen (that is, the crosshairs), you must create an offset for the horizontal and the vertical separately.

The resulting offset draft line is not associative to the axis line or datum. This means that when the axis or datum is clipped, moved, or erased, the draft line remains unchanged.

The new draft line is automatically related to the view. When you offset a draft entity from a model edge or another draft entity already related to a view, the system automatically relates it to the view from which you selected the model edge or the other draft entity.

1. Choose DRAWING **> Tools > Offset** > **Single Ent**.
2. Select either a parallel or a perpendicular axis.
3. Choose the attributes of the draft line from the OFFSET TYPE menu and choose **Done/Return**.
4. Enter the offset distance.
   When you choose **Single Ent** from the OFFSET OPER menu, the OFFSET TYPE menu displays the following commands:
   - **Fixed**—Creates a draft entity with a single offset value.
   - **Tapered**—Creates a draft entity with a different offset at each endpoint.
   - **Trimmed**—Offsets only a portion of the draft entity. Determine the trimmed boundaries by selecting a draft point on the entity, or by selecting some arbitrary point on the entity.
   - **Untrimmed**—Creates a draft entity that uses the whole length of the original draft entity.

Using the **Ent Chain** command in the OFFSET OPER menu, you can simultaneously select several entities to offset. The entities must all be connected, but you can select them in any order or use the **Pick Box** command.

Note: The **Ent Chain** command is not available for selecting axes.

Example: Creating Draft Entities Using Offset
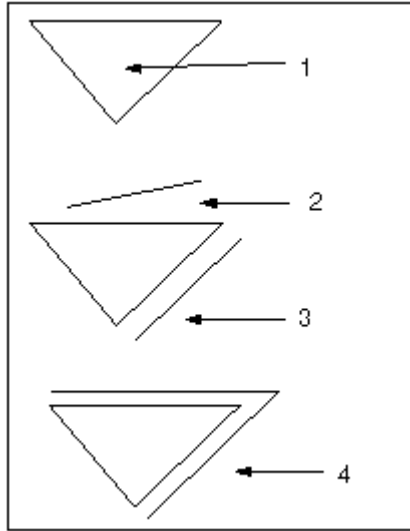
1   Original draft entities.
2   Created using Single Ent, Tapered, and Trimmed.
3   Created using Single Ent, Fixed, and Untrimmed.
4   Created using **Ent Chain**.

About the Translate Command

Using the **Translate** command in the TOOLS menu (DRAWING **> Tools > Translate**), you can move draft entities and annotations about the drawing if they are not attached to an edge. The **Translate** command differs from the **Move** command in the following ways:

- You can translate any number of items simultaneously.

- Once you have translated the selected entities, the system unselects them automatically. Using the **Move** command, you can move the item repeatedly until you press the middle mouse button.

  Note: You cannot use **Translate** in the TOOLS menu to copy entities as you would use **Copy** in the TOOLS menu, followed by **Translate** in the COPY TYPE menu.

To Translate Draft and Unattached Entities

1. Choose DRAWING **> Tools > Translate**.
2. Select draft entities to translate. Choose **Done Sel** when you have finished.
3. Choose a command from the GET VECTOR menu to specify the direction for translating the new entities, and then follow the system prompts. The system uses the specified distance values between each copy of the draft geometry.

To Move Draft Entities to Another Sheet

Using the GET SELECT menu, you can select draft items that are not associated with any view and move them to another sheet. The system moves draft items that are associated with a view to another sheet if you switch the view to which they belong to that sheet.

To Break a Draft Entity

1. On the menu bar, click **Insert > Break**. The BREAK menu opens in the Menu Manager. **Add** is the default option.
2. Select an entity.
3. Indicate breaks by selecting the start point and endpoint of each break.
4. After the system breaks a draft entity into two separate draft entities, select the first break point on either of the two entities to start another break.

To Stretch a Draft Entity

1. Choose DRAWING > **Tools** > **Stretch**.
2. Use the left mouse button to specify two opposite corners of a selection box within which to enclose the entities you want to stretch.
   Note: Filled draft entities cannot be stretched.
3. To edit your selection, use the GET SELECT menu to select individual entities that you want to exclude from the selection and press the middle mouse button.
4. Define the translation vector by selecting the first location—the *from* location; then select the final destination point. The specified entities are stretched.

Stretching Draft Entities

The stretching process does not affect the location of entities outside the selection box.

Lines that cross the selection box stretch so that the system translates their location point within the box along the translation vector.

The stretching process does not affect circles and arcs with centers outside the box.

The system does not preserve tangency in the process of stretching.

To Divide a Draft Entity

1. Click **DRAWING > Tools > Divide.**
2. Select the entities to divide, and then type the number of equal segments into which you want to divide the entity. If you select multiple entities, the system divides them all the same way into equal segments. After the entity divides, it highlights the break points.

To Change the Line Style of Draft Entities

To change the line style of selected entities as well as redefine some or all line style elements such as line font, color, and line width (thickness):

1. In the menu bar, click **Format > Line Style**.
2. Choose one of the following options in the LINE STYLES menu:
   - **Modify Lines**—Allows you to select draft entities whose line styles you want to modify, using the **Modify Line Style** dialog box.
   - **Edit Styles**—Opens the **Line Style Library** dialog box, from which you can create a new line style, edit an existing style, or delete a style.
   - **Edit Fonts**—Opens the **Line Font Library** dialog box, from which you can create a new font, edit an existing font, or delete a font.
   - **Clear Style**—Undoes the last line style change you made.

The Rescale Command

Using the **Rescale** command in the TOOLS menu (DRAWING **> Tools > Rescale**), you can modify the size of draft geometry. When the system regenerates the drawing, it updates associative dimensions to the resized

geometry. However, you must scale nonassociative dimensions with the geometry to maintain their relative positions. These values do *not* change with scaling.

To Scale Draft Geometry

1. Choose DRAWING **> Tools > Rescale**.
2. Select draft geometry and dimensions to scale, and then click **Done Sel**.
3. Use the GET POINT menu commands to select the point about which to scale the entities.
4. Type a scale value and press ENTER. A number greater than 1 makes the entities larger; a number less than 1 makes them smaller. For example, a value of .25 scales the drawing to one-quarter (1/4) of the actual size of the model.
5. Regenerate, if necessary, to recalculate associative dimensions.

To Modify the Diameter of an Arc or Circle

When you modify the diameter of an arc or circle, the system does not update unassociated draft dimensions to reflect changes in the entity size.

1. Select the draft entity (circle or arc) whose diameter you want to modify, and then right-click to display the shortcut menu.
2. Choose **Mod Arc Diam** from the shortcut menu.
3. Type a new diameter for the draft entity. The entity reappears at its modified value.
4. To update any associated draft dimensions, choose DRAWING > **Regenerate** > **Draft**.

Using a Model Edge

Using the **Use Edge** command in the TOOLS menu (DRAWING **> Tools > Use Edge**), you can create a draft entity using a model edge, and the system automatically relates it to the view. When you erase the view, the system erases all attached detail items with it. When you delete the view, the system deletes all attached detail items; however, you can use **Relate View** in the VIEWS menu (DRAWING **> Views > Relate**) to remove the draft items and make them independent of the view.

To Create a Draft Entity Using a Model Edge

1. Choose DRAWING **> Tools > Use Edge**.
2. Select a model edge and choose **Done Sel**.
3. Type [Y] to erase the selected edge. Regenerate the view to redisplay the selected view (**View > Update** on the menu bar).

About Modifying Splines

In Drawing mode, you can modify splines in the following ways:
- Move a single point or a range of points on the spline
- Add points to the interior of the spline, or extend the spline by adding points to the exterior
- Delete points from the spline
- Use a deviation value to reduce the number of spline points
- Smooth the spline
- Use the spline control polygon to modify the spline

To Move a Single Spline Point

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus appear in the Menu Manager.
3. To move a single point on the spline, click **Single Pnt**, and then select the point you want to move. The

point and the part of the spline that is affected moves as you move the mouse pointer to another location on the screen.

4. Click in the new location to place the point.
5. Click **Done Modify** when you are finished. To cancel the move operation, click **Quit Modify**.

To Move a Range of Points on a Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus appear in the Menu Manager.
3. Select two spline points limiting the adjustment range.
4. Select the point (within the adjustment range) you want to move.
5. To place the point in a different location, click another location on the drawing window. The selected point moves to its new location, and the part of the spline enclosed within the range adjusts accordingly.

To Add Points to a Spline

You can add points to the interior of a spline, or to the exterior, thus extending the spline.

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus open in the Menu Manager.
3. In the MOD SPLINE menu, click **Add Points**. **Interior** is selected by default in the NEW POINTS menu.
4. Do one of the following:
   – Choose **Interior** and add points to the interior of the spline by selecting any location on the spline between any two existing points.
   – Choose **Exterior** and extend the spline by adding points beyond its current endpoints. Select the spline endpoint to extend; then select additional points.

To Modify a Spline Using Control Poly

You can modify a spline by using its control polygon (the polygon that the system creates to surround the spline when you first create it). The control polygon displays in white when you choose the **Control Poly** command.

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus open in the Menu Manager.
3. Choose MOD SPLINE > **Control Poly**.
   The spline's control polygon appears in white. Spline control points appear with line segments between them on the spline. The line segments begin and end at the startpoint and endpoint of the spline, and intermediate segments remain tangent to the spline. These line segments are visual aids for modifying the shape of the spline.
   Note: You cannot use this command on a spline that has a tangency condition defined for one end only.
4. Adjust the shape of the spline by selecting a point on the control polygon and dragging it to a new location. You *cannot* select endpoints. The spline rubberbands to its new shape as you move the point.
5. Place the control point by pressing the left mouse button.

To Delete Points from a Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus open in the Menu Manager.
3. Choose MOD SPLINE > **Delete Points**.
4. Select a spline point to delete. Click **Done Sel**, then **Done Modify**, when you are finished removing points. The system redraws the spline according to the new shape created when you removed the points.
5. Repaint the drawing to see the new spline shape.

To Decrease the Number of Spline Points Using a Deviation Value

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus open in the Menu Manager.
3. Choose MOD SPLINE **> Sparse**.
4. Type a deviation value to redraw the spline with fewer points. This value must be a positive number. The system highlights the spline and tells you how many points it is going to remove.
5. To accept the changed spline, choose **Accept**; to reject it, choose **Reject** and type a different deviation value. (For each spline, you might need to try several different deviation values before achieving the desired result.)

To Smooth the Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Choose **Modify Spline** from the shortcut menu. The MOD SPLINE and MOVE PNTS menus open in the Menu Manager.
3. Choose MOD SPLINE > **Smooth**.
4. Type the number of points to average in order to smooth the spline:
   - Type [1] to keep the spline as is.
   - Type [3] to average the centermost point out of the group and one point on each side of it.
   - Type [5] to average the centermost point in the group and two points on each side of it.
5. The resulting spline appears in green. To accept it, choose **Accept**; to reject it, choose **Reject** and try again with a different number of points.

To Move or Clip Items in Drawing Mode

You can move items from one location to another in the drawing window. You can move individual objects as well as entire views. You can also extend or shorten the length of lines and other boundaries. The moving and clipping features in Pro/ENGINEER work differently when you are in Drawing mode than when you are in Part or Assembly mode. To move or clip an item while you are in Drawing mode:

1. In the drawing window, select the item. The item is highlighted and contains handles (boxes).
2. Do one of the following:
   - To clip or extend a line, click a handle, and then click anywhere outside of the item to extend or shorten its boundary.
   - To move an item, click anywhere in the drawing window. The item moves to the selected location.

About Controlling the View Type

You can control the extent to which the model is visible in a drawing by creating full, half, partial, and broken views.

As you specify a major view type, you can use the commands in the VIEW TYPE menu to control how much of the model is visible in the drawing:

- **Full View**—Shows the model in its entirety.
- **Half View**—Removes a portion of the model from the view on one side of a cutting plane.
- **Broken View**—Removes a portion of the model from between two selected points, and closes the remaining two portions together within a specified distance.
- **Partial View**—Displays a portion of the model in a view within a closed boundary. The system displays the geometry appearing within the boundary, but removes the geometry outside of it.

Creating Half Views

When Pro/ENGINEER creates a half view in a drawing, it cuts the model at a plane, erasing one portion of it, and displaying the rest. The cutting plane may be a planar surface or a datum, and must be perpendicular to the screen in the new view. The **Half View** command is available only for projection, auxiliary, and general views.

Example: Half View
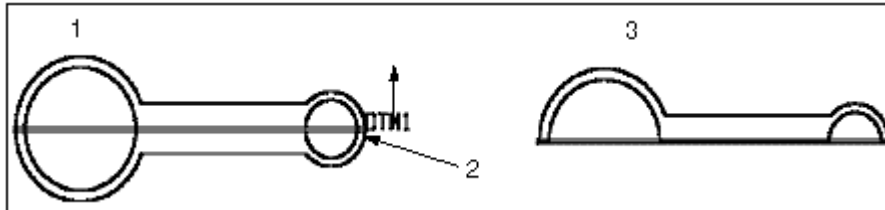
1   The view temporarily appears in full.
2   Select this datum plane as the cutting plane.
3   The resulting half view.

To Create a Half View

1. Follow the procedure for adding a projection, auxiliary, or general view; then choose VIEW TYPE > **Half View**.
2. Select the cutting plane from any other view on the drawing (it *must* be perpendicular to the screen in the new view).
3. Arrows appear on the new view pointing to the side that is to remain in the view. Choose **Flip** to flip the direction of the arrow or **Okay** to accept the direction and continue.
    The view that was on the screen changes. The system discards the portion of the view that was behind the cutting arrows, and the cutting arrows do not remain in the drawing.

Creating Partial Views

To create a partial view, you define and orient a view, and then sketch a closed boundary around the new view. The system displays the geometry within the closed boundary, but not geometry outside of it. The **Partial View** command is available only for projection, auxiliary, and general type views. You can scale them, cross-section them, and create them with several boundaries or breakouts to include a local area cross section.

Example: Partial View
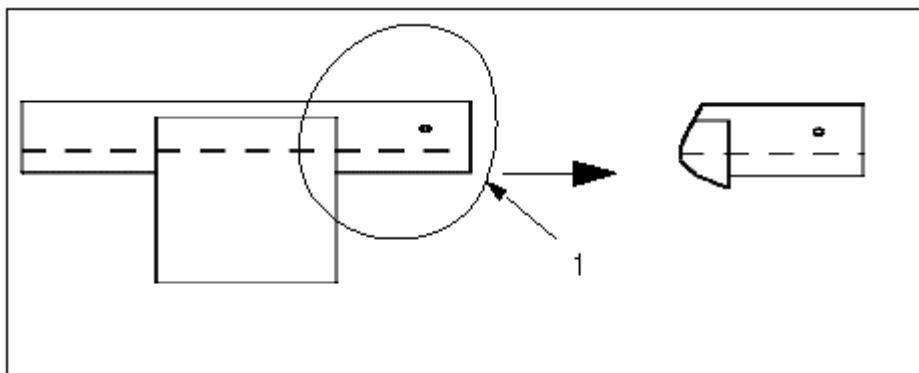
1   Sketch this spline to define the view boundary.

To Create a Partial View

1. Follow the procedure for adding a projection, auxiliary, or general view; then choose VIEW TYPE > **Partial View**.
2. Select a point on an edge in the new view. A red cross appears, representing the reference point on the geometry to be displayed in the new view. The system needs this reference point to properly regenerate the partial view.
3. Draw a boundary around the red cross, enclosing all of the geometry that you want to display in the partial view. Select points using the left mouse button. To close the boundary, press the middle mouse button. Only the portion of the view that was inside the spline appears.

Creating Broken Views

You can create broken views in both vertical and horizontal directions with as many breaks as you want. You can also create broken views with local cross sections.

The space between the breaks—the offset distance—is defined as a vertical or horizontal distance between two subsequent pieces of the view. The display of a broken view maintains a constant offset distance between the displayed pieces of the view when you change the following:

- Model geometry
- Scale of the drawing
- Reference points of the broken view

You can control the offset distance when you first create a break by setting the drawing setup file option `broken_view_offset`. The default spacing is 1 drawing unit. You can change the spacing by using **Move View** to move one of the subviews, or portions, of the broken view.

You can draw the break line for a broken view in three ways:

- **Sketch**—Sketch a spline
- **S-Curve**—An S-Curve shaped spline is created for the break line
- **Heartbeat**—A heartbeat-shaped spline is created for the break line

When you select **S-Curve** or **Heartbeat** to use as your spline type, the following two options are available:

- **View Outline**—Scales the spline based on the view outline in that direction
- **Geometry**— Scales the spline based on the distance between two selected points

You can create a projection view from a broken view. The break points are projected, allowing you to draw new splines in the child view.

Note: The shape of the S-curve and Heartbeat is invariant; that is, any change in size affects both length and width proportionally.

To Create a Broken View

1. Choose **Projection** or **General** and **Broken View** from the VIEW TYPE menu.
2. Select the location for the center of the new view. The object appears. Orient the view if necessary.
3. For each break in the view, define the boundaries of the vertical or horizontal area to remove. Choose **Add** or **Delete** from the ADD/DEL BRK menu, or choose **Vertical** or **Horizontal** from the BREAK DIR menu.
   Note: You cannot add horizontal break lines on a horizontally projected broken view, or add vertical break lines to a vertically projected broken view.
4. To add a break line, select on a model edge. For every two break lines that you add, the system creates one break and removes the portion of the view that is between them.

View with Break Lines
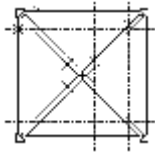
5. When you have finished adding break lines, choose ADD/DEL BRK > **Done**. The system removes the area between break lines and displays the model, indicating the break by a single horizontal or vertical line.

View with Area Between Break Lines Removed
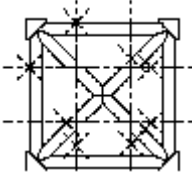
6. Sketch the spline break representations. Select all breaks in the same direction that are to use the same spline. You must sketch vertical and horizontal breaks separately.

Sketch of Spline Break Representations
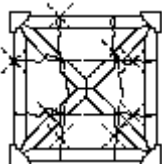
7. Choose **Done Sel**; then sketch the spline. Do this for every break that has a different spline representation. If you do not sketch a spline for a break, the break representation is in a straight line. For multiple break lines with a single sketch, sketch the break line at the location of the first break selected; otherwise, the break lines all shift.
8. Choose **Done** when you have finished. The view expands at the breaks with the contours just sketched and an initial gap between each portion (determined by the drawing setup file option "broken_view_offset").

Completed Broken View
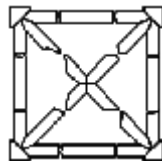
To Add a Segment to a Broken View

1. Click **Views > Modify View > Boundary**.
2. Select the view you want to redefine. The view segments temporarily snap together.
3. On the VIEW BREAK menu, click **Add Segment**. The view segments return to their previous positions.
4. Select a view border to which you want to create a new segment. Doing this creates the new broken view segment. You can repeat this step as many times as you want to create additional segments.
    Note: The new broken view segment is created in the view and any appropriate projected views. The new segment has the same reference points and spline shapes as its neighbors, and it is offset at the same distance.

To Delete a Segment from a Broken View

1. Click **Views > Modify View > Boundary**.
2. Select the view you want to redefine. The view segments temporarily snap together.

3. On the VIEW BREAK menu, click **Del Segment**. The view segments return to their previous positions.
4. Select the segment you want to delete. The segment is highlighted.
5. Select a side of the highlighted segment where you want to keep the spline shape. Doing this deletes the segment.
   Note: The deleted segment is deleted in the current view and any appropriate projected views.

Tip: Moving Broken Views

When you move a broken view, for any subview (or portion of the view) that you select to move, all subviews to its right and below it move the same distance. To move the entire broken view to a different location on the drawing, select the upper-left subview. This moves the entire view without altering the gaps between the subviews. Selecting any other subview moves all subviews below it and to the right of it the same distance.

Moving a Broken View
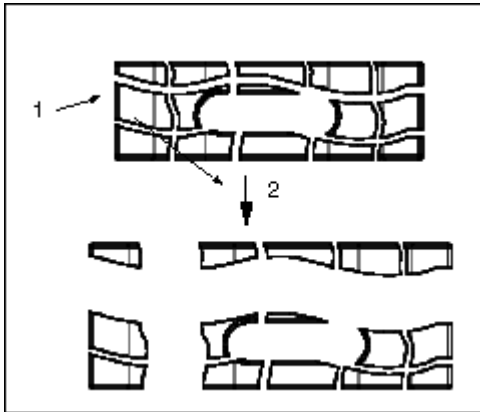
1   Select this subview to move the view as a whole.
2   Move vector.

About Exploded Assembly Views

When you are making a drawing of an assembly, you can add an exploded view of the assembly without having to explode it in Assembly mode. If you save an assembly in its exploded state in Assembly mode, and then retrieve the view in Drawing mode by selecting a name from the **Saved Views** list in the Orientation dialog box, the system places the view in the proper orientation, but not in its exploded state.

To Create an Exploded View

1. Choose **Views > Add View**.
2. Choose commands from the VIEW TYPE menu, **Exploded**, and **Done**.
3. Select the center of the view.
4. Specify the orientation type using the Orientation dialog box. The system adds the exploded assembly view to the drawing.
   Note: If you have a license for Pro/PROCESS for ASSEMBLIES, the SEL STATE menu appears. You can then select an explode state and orient it.

Creating Exploded Views

You can create multiple exploded views in the same drawing. When you create an exploded view using the **Add View** command, Pro/ENGINEER copies current exploded dimensions of the model into the view. After that, the explosion distances of that view are independent of the other views and of the model itself.

If you have modified the **Expld Status** values of an assembly in Assembly mode (by switching them on and off), the system may not show more than one exploded view. You can control exploded dimensions in Drawing

mode, but to control components globally, use the **Expld Status** command in Assembly mode.

Changing Explosion Distances for Exploded Views

The explosion values specified in the assembly control the explosion distance between components in the model. You can modify exploded dimensions in a drawing view by choosing **Mod Explode** from the VIEW MODIFY menu. However, this command only alters the drawing cosmetically and does not modify the actual exploded dimensions of the model. If you modify an assembly in Assembly mode by choosing **Redefine** from the EXPLD STATE menu, you cannot change these explosion parameters in a drawing.

To Modify the Explosion Distances Between Components

1. Click **VIEWS > Modify View**. The **VIEW MODIFY** menu appears.
2. Click **Mod Expld** and then select a view to modify.
3. Choose EXPLD STATE > **Redefine** > **Position**.
4. Change the explosion distances by using commands in the MTNPREF menu.
   Note: If you do not have a license for Pro/PROCESS for ASSEMBLIES, the MOD EXPLODE menu appears instead of the EXPLD STATE menu. You can choose **Position** to access the MTNPREF menu, or **Explode Status**.

When you choose **Exploded** from the VIEW TYPE menu, you must specify which explode state to use by selecting it from the SEL STATE menu. If the view that you want to use is a process model view, you must select the explode state from the SEL STATE menu.

Creating Single-Surface Views

If you have a Pro/DETAIL license, you can create single-surface views out of a solid surface or datum quilt. Pro/ENGINEER always displays them in wireframe, but it does not display datums, cosmetic features, and coordinate systems. It erases all geometry other than the selected surface. You can create regular projection views as well as single-surface projection views from single-surface views.

Example: Single Surface Views
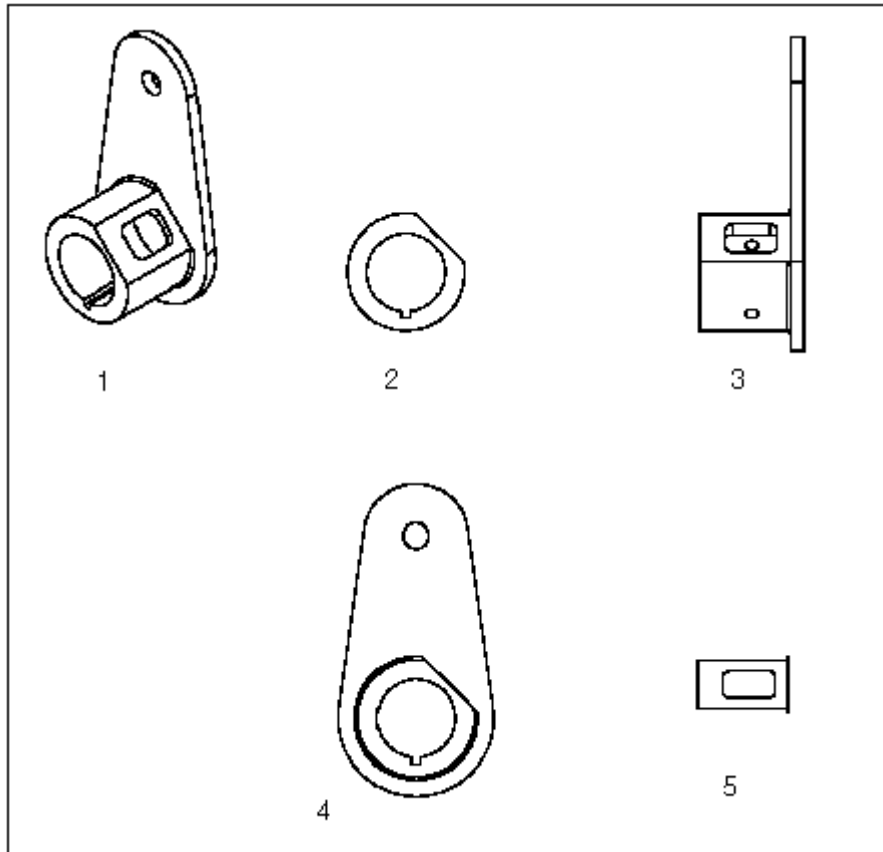
1   General Full view.
2   General Single-Surface view.
3   Projection Full view.
4   General Full view.
5   Projection Single-Surface view.

To Create a Single-Surface View

1. Click **VIEWS > VIEW TYPE > Of Surface** and any other commands except **Detailed**.
2. If the single-surface view is **General**, orient the model as usual.
3. Select the single part surface to show. The system erases all other geometry.

About Perspective Views

You can create a perspective view of a model in a drawing by specifying an eyepoint distance, or *focal length*, and a view diameter.

Creating Perspective Views

When creating and modifying perspective views, keep in mind the following restrictions:

*   You can display only general views in perspective.
*   Perspective views cannot have cross sections.
*   You cannot use perspective views for creating projection or detailed views.

- Dimensions, datum planes, and axes do not appear in perspective views (except when orienting the view).
- You cannot use the **Intersect** and **Midpoint** attachment commands to attach items with leaders to perspective views.
- In Part mode, if you create a **From/To** perspective view and then name it (by typing a name in the **Name** box of the Orientation dialog box), you cannot use this named view in a drawing. The system places it in the default view.

Example: Perspective View
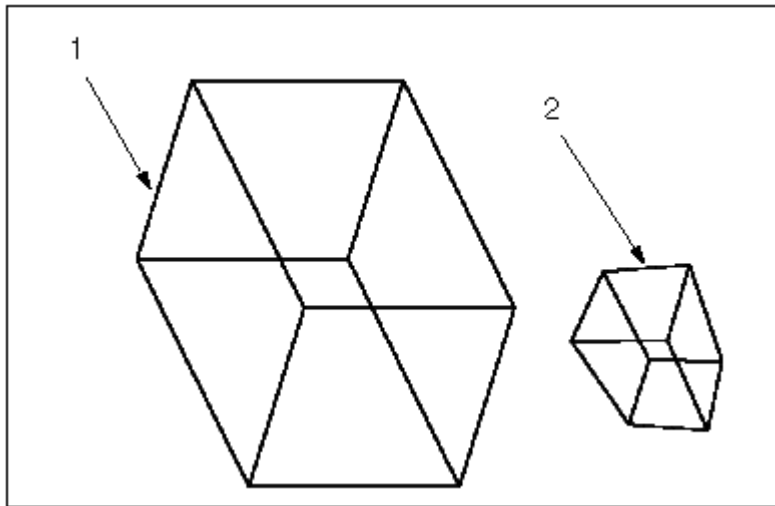
1   General view
2   Perspective view. The view diameter is 1.5000. The eyepoint distance is 20.000.

To Create a Perspective View

1. Click **VIEWS > Add View > General > Full > No Xsec > Perspective > Done**.
2. Select a point to locate the view.
3. Type the eyepoint distance in model space units (inches).
4. Type the view diameter in paper units (inches).
5. The general view appears in perspective. Orient the view using commands in the Orientation dialog box; then choose **Done/Return** to finish.

To Modify a Perspective View

1. Click **VIEW MODIFY > Perspective**.
2. Select the view on the screen.
3. Using the MOD PERSPECT menu, do one or both of the following, as necessary:
   - Change the eyepoint distance by choosing **Viewing Dist** and entering a value.
   - Change the diameter by choosing **Diameter** and entering a value.

About Cross-Sectional Views

You can create a cross section in Part and Assembly modes and show it in a drawing or you can add it to a view in Drawing mode while you are creating it. If you have a license for Pro/DETAIL you can use commands in the XSEC TYPE menu to create ten types of cross sections:

- A *full* cross section displays the whole view, whereas a *local* cross section shows a portion of the model within a closed boundary cross-sectioned, but not outside the closed boundary.
   - A *full & local* cross section shows a full cross-sectional view with local cross sections.

- A *half* cross section shows a portion of the model on one side of a cutting plane, but not on the other side.
- A *total* cross section shows not only the cross-sectioned area, but the edges of the model that become visible when a cross section is made.
- An *area* cross section displays only the cross section without the geometry.
- An *aligned* cross section displays an area cross-sectional view that is unfolded around an axis, whereas a *total aligned* cross section shows an aligned cross section of a general, projection, auxiliary, or full view.
- An *unfolded* cross section shows a flattened area cross section of a general view, whereas a *total unfolded* cross section shows a total unfolded cross section of a general view.

Assigned and Displayed Crosshatching Patterns in Cross-Sectional Views

When creating a drawing view that contains a cross-section of the associated model, keep in mind that assigned crosshatching patterns for model cross-sections may be based on the assigned material of the part. For example, if you created a cross-section of a part or assembly and the cross-section cuts through a part that has a defined material (such as steel), the system looks for a crosshatching pattern that has the same name as the assigned material (without the extension .xch). If the system finds such a pattern, it automatically assigns it instead of the standard default crosshatching pattern to the cross-section. The assigned crosshatching pattern is displayed in Part and Assembly modes, and also in Drawing mode when you create the cross-sectional view.

If the cross-section cuts through a part that does not have a defined material, the system assigns the default crosshatching style to the cross-section. Again, the default style displays in Part, Assembly, and Drawing modes.

Note: If either of the two parameters `default_xhatch_angle` and `default_xhatch_spacing` are defined for the part containing the cross-section, these parameter settings supercede other definitions of the crosshatching style. That is, the existence of an assigned material name is no longer relevant to the assignment of the crosshatching style.

To Include Surface Geometry in Cross-Sectional Views

Include surfaces in drawing cross sections by changing the `no` value of the `show_quilts_in_total_xsecs` detail setup file option to `yes`. Exclude surfaces by changing the `yes` value back to `no`.

Surface geometry (surfaces and surface quilts) will only be cut by the cross-section cutting plane in a drawing view when the cross-section is created as a Model & Qlts or Surf/Quilt cross-section in part or assembly mode. If the cross-section is created as a Model (default), the surface geometry will not be cut by the cutting plane.

Full Cross-Sectional Views

When you create a view, you can retrieve any of the existing cross sections or create a new cross section *on the fly*. To control the display of datum curves, threads, cosmetic feature entities, and cosmetic crosshatch in a full cross-sectional view, use the drawing setup file option `remove_cosms_from_xsecs`. If you set it to `all`, you can remove datums and cosmetics from all types of cross-sectional views.

To Create a Full Cross-Sectional View

1. Click **VIEW TYPE** > **Section** and then **Done**.
2. Choose the cross-section type from the **XSEC TYPE** menu and then click **Done.**
3. Using the **XSEC ENTER** menu, do one of the following:
   - Choose **Create** and make a cross section on the fly.
   - Choose **Retrieve** and select a cross section from the XSEC NAMES menu.
4. Select **Planar** from the XSEC ENTER menu to create a planar cross section or select **Offset** to create an

offset cross section.

5. Choose **Done** and type a name for the cross section.
6. Create a section.
7. Continue with the process of creating a view.

Half Cross-Sectional Views

Using the **Half** command in the VIEW TYPE menu, you can create a half cross-sectional view. The portion of the model cross-sectioned depends on the direction of the cutting arrow. The cutting arrow's direction is reversible.

Example: Half Cross-Sectional View
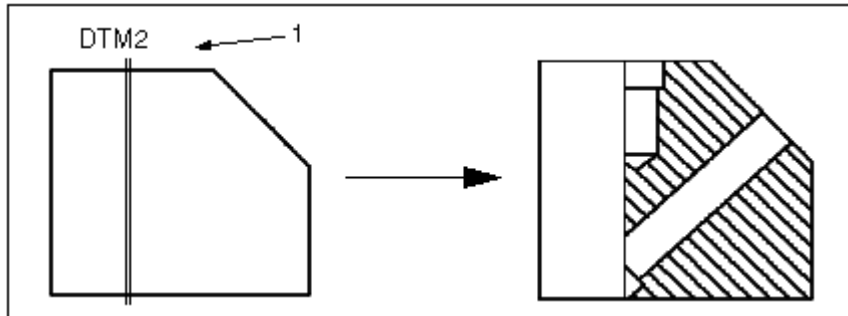
1   Select the datum plane as the boundary for the cross section.

To Create a Half Cross-Sectional View

1. Select the view type from the VIEW TYPE menu.
   Note: Half is *only* available if you choose **Full View** from the VIEW TYPE menu. This command is *not* available for partial, broken, or half views.
2. Choose XSEC TYPE > **Half** > **Done**.
   Note: The XSEC TYPE menu appears only if you have Pro/DETAIL. Otherwise, the system selects **Area Xsec** automatically.
3. Orient the view as desired; then select a reference plane for a half cross section.
4. Arrows show the direction of creation of the half cross section. Confirm the direction by choosing **Okay**, or reverse it by choosing DIRECTION > **Flip**.
5. Select or create a cross section using commands in the XSEC ENTER menu.
6. Select the view to show cross-section arrows, or press the middle button for none.

Local Cross-Sectional Views

You can create a *partial* projection, auxiliary, general, or revolved view with breakouts for a local cross section. You can create local cross sections as you create a view, or you can add them later by modifying the view. In both cases, the VIEW BNDRY menu provides the following commands to add, delete, or modify local cross sections:

- **Mod Breakout**—Modifies boundaries of a local cross section

- **Add Breakout**—Adds a local cross section

- **Del Breakout**—Deletes a local cross section

- **Move Ref Pnt**—Relocates a reference point of a local cross section
  Note: You cannot create nested breakouts in views with local cross sections.

To Create a Local Cross-Sectional View

1. Choose VIEW BNDRY > **Add Breakout**.
2. Indicate a cross section to use for a local cross section by selecting one of the following from the NEW BREAKOUT menu:
   - **Current Xsec**—Uses the cross section that was referenced by the previously created cross section.
   - **Choose Xsec**—Creates a cross section on the fly, or retrieves an existing one. Use commands in the XSEC ENTER menu.
3. Select a view to show arrows or press the middle mouse button for none. (You need not perform this step if you choose **Current Xsec**.)
4. Select a reference point on the view geometry to be the center for a local cross section.
5. Sketch a spline without intersecting other splines. Finish the spline by pressing the middle mouse button. To finalize cross sections that you created, choose **Done**.

Example: Local Cross-Sectional View
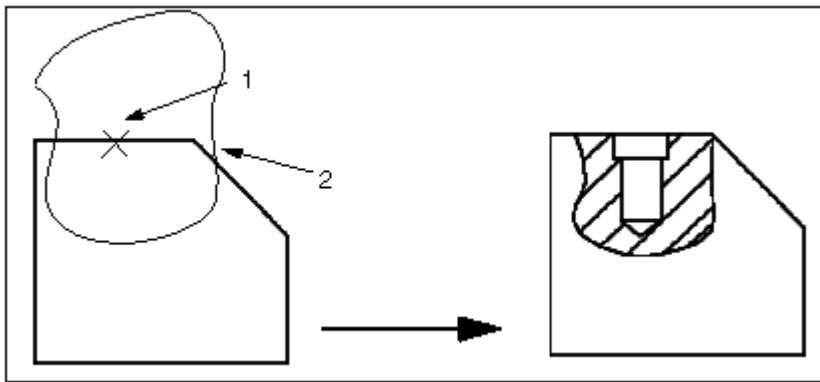
1 Select on the edge to locate the center of the local cross section.
2 Sketch this spline to define the cross-section boundary.

To Create a Partial or Broken View with Local Cross Sections

1. From the VIEW TYPE menu, choose **Partial View** or **Broken View**.
2. Choose **Section** > **Done** > **Local**.
   Note: Local is *not* available for half views. It is available *only* if you choose **Full View** or **Partial View** from the VIEW TYPE menu.
3. Select a center point for the view and orient the view as desired.
4. Create or retrieve a cross section for a local cross section to reference. Use commands in the XSEC ENTER menu.
5. Select a view for arrows or press the middle mouse button for none.
6. Do one of the following:
   - If you are creating a partial view, select a center point for the partial view, sketch a spline defining partial view boundaries, and select a center point for a breakout. To create more local cross sections, specify a cross section to reference and then define the center and boundaries of the breakout. Choose VIEW BNDRY > **Done**.
   - If you are creating a broken view, select a center point for the breakout and follow Steps 7 and 8.
7. Add breaks by selecting reference points and choose ADD/DEL BRK > **Done**.
8. Select lines to share the same spline; then sketch a spline.
9. Choose EXIT > **Done**.

Example: Partial View With Local Cross-Sections
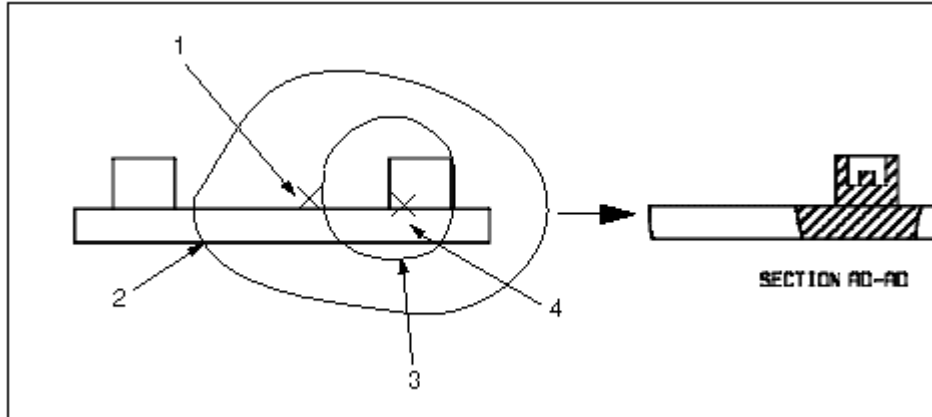
1 Select this point for the center of the partial view.
2 Sketch this spline to define the view boundary.
3 Sketch this spline to define the boundary of the local cross section.
4 Select this point for the center of detail.

To Create a Full Cross-Sectional View with Local Cross Sections

1. From the VIEW TYPE menu, choose **Projection** or **Auxiliary**; then choose **Full View** or **Partial View**.
   Note: This view type can be a total cross-sectional view *only*.
2. Choose Section > Done > Full & Local.
   Note: Full & Local is *not* available for broken and half views.
3. Select a center point for the view and orient the view as desired.
4. Select or create a cross section using commands in the XSEC ENTER menu.
5. Select a view for arrows or press the middle button for none. The system creates a full cross section.
6. For a partial view, create an outer boundary by picking a reference point and sketching the boundary.
   Note: Boundaries and reference points of local cross sections appear in orange; outer view boundaries and reference point appear in green.
7. Specify another cross section for the local cross section to reference by choosing one of the following from the NEW BREAKOUT menu:
   – **Current Xsec**—Uses the current cross section for creating a local breakout. This command is valid if you want to choose the cross section that was already referenced by a previously created local cross section.
   – **Choose Xsec**—Specifies a cross section for the a local cross section to reference. Create or retrieve one using commands in the XSEC ENTER menu.
8. Select a view for arrows or press the middle mouse button for none.
9. Select a center point for the local cross section.
10. Sketch a spline without intersecting other splines. The spline appears in orange.
11. If desired, create more local cross sections by first specifying a cross section to reference and then defining the center and boundaries of a breakout.
12. Choose VIEW BNDRY > **Done** when you have finished.
13. Modify the crosshatch pattern for local breakouts so that it differs from that of the full cross section.

Example: Full Cross-Sectional View with Local Cross-Sections
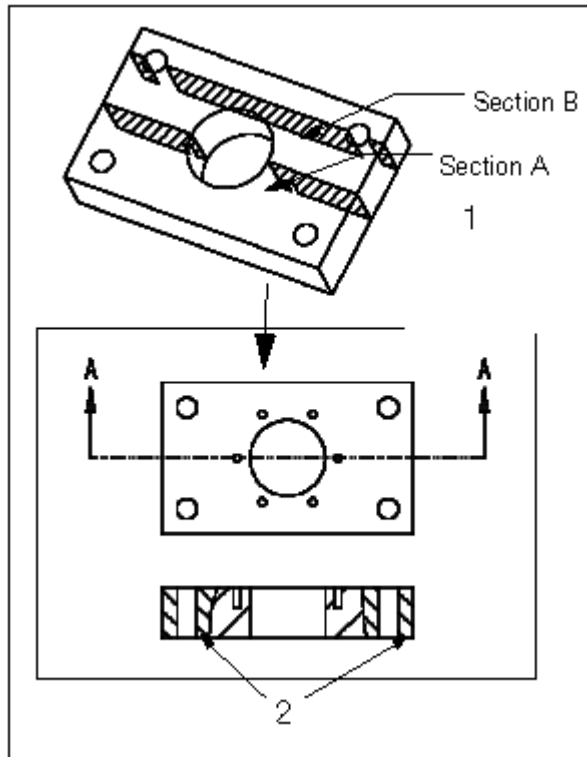
1   Created a full cross section using section A.
2   Created breakouts for the local cross sections by referencing section B. Modified crosshatch after the view was created.

Total and Area Cross-Sectional Views

A *total* cross section shows not only the cross-sectioned area, but the edges of the model that become visible when a cross section is made, whereas an *area* cross section displays only the cross section without the geometry. Area cross sections display only geometry that lies in the cutting plane. Cosmetic features, datums, and coordinate systems do *not* appear. Quilts do not appear in total cross sections; however, they *do* appear in area cross sections created using the **Model & Quilts** command. In planar area cross-sectional views, you can show all cosmetic sketches and datum curve features that lie in the cutting plane by setting the drawing setup file option `draw_cosms_in_area_xsec` to yes.

To Create a View with a Total Cross Section or an Area Cross Section

1. Add a view using General, Section, Full, and Total Xsec or Area Xsec.
    Note: Total Xsec or **Area Xsec** are *only* available if you choose **Full** from the XSEC TYPE menu.
2. Select the location for the view.
3. Select or create a section using commands in the XSEC ENTER menu.
4. Select the view for showing arrows and set the direction of viewing by choosing DIRECTION > **Flip** > **Okay**. If you do not select a view to show arrows, press the middle mouse button.

Examples: Total and Area Cross-Sectional Views
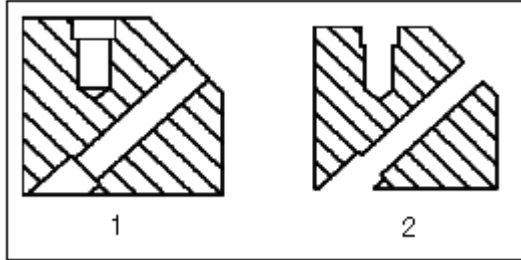
1   Total cross section.
2   Area cross section.

Aligned Cross-Sectional Views

An *aligned* cross section displays an area cross-sectional view that is unfolded around an axis. The system revolves all cutting planes of an offset cross section about the selected axis until they are oriented parallel to the screen (sheet). Aligned cross sections create area-type cross sections. A *total aligned* cross section shows an aligned total cross section of a general, projection, auxiliary, or full view. You can make a detailed view from an aligned cross-sectional view.

The following restrictions apply to both aligned cross-sectional views and total aligned cross-sectional views:

- You can align offset cross sections only.
- The axis around which the cutting planes revolve must be parallel to the screen.
- The axis must lie on all cutting planes.
- The axis must be coaxial with all cylinders in the section.
- In an aligned cross section, you cannot show model dimensions. Dimensions can be created within one segment of the cross section.
- If the view is to be general, orient the model so that the axis around which you are going to unfold is parallel to the screen.
- Pro/ENGINEER creates aligned and unfolded cross-sectional views as area cross sections. No crosshatch appears unless you have a Pro/DETAIL license.

To Create an Aligned or Total Aligned Cross-Sectional View

1. From the **VIEW TYPE** menu, choose **Projection**, **Full View**, **Section**, **No Scale**, and **Done**.
2. From the **XSEC TYPE** menu, choose **Full**, **Align Xsec** (or **Total Align**), and **Done**.
   Note: The **Align Xsec** and **Total Align** commands are only available if you choose **Full** from the XSEC TYPE menu.
3. Select a center point for the view.
4. Select or create an offset section to use to unfold the view. Use commands in the XSEC ENTER menu.
5. Select the axis about which the cutting planes is to revolve to be parallel to the screen. Follow the applicable restrictions.
6. Select a view that displays arrows designating the viewing direction and the cutting plane outline. Press the middle button for none.
7. Choose **Flip** and **Okay** to change or confirm the direction of viewing arrows.

Unfolded Area Cross-Sectional Views

You can create an unfolded cross section or a total unfolded cross section of a general drawing view by unfolding (stretching) cutting planes until they become parallel to the screen.

When you are unfolding a cross section, the following restrictions apply:

- You can only unfold offset cross sections that were created using the **Both Sides** command.
- The entities that you use to define the cross section must be lines.
- You cannot create a projected view from an unfolded cross-sectional view.
- Pro/ENGINEER does not support part dimensions in unfolded cross section drawing views.
- You cannot create partial and half total unfolded views.
- You cannot create projection views of total unfolded views.
- You can show dimensions of features in the view if each dimension stays within a single segment of the section.
- If you deleted the offset section of a total unfolded cross-sectional view from the model or you cannot regenerate it (such as when the sketching datum plane is suppressed), the total unfolded cross-sectional view becomes a regular general view.

You can control the display of seams (the edges of the cutting plane) in total unfolded cross-sectional views by using the drawing setup file option `show_total_unfold_seam`. When you set it to `yes`, the seams are visible. When you set it to `no`, the system blanks them. To update the view display after you reset the drawing file setup option, choose **Update > Drawing View** from the Pro/ENGINEER **View** menu.

To Create an Unfolded or Total Unfolded Cross-Sectional View

1. Click **VIEWS > Add View**.
2. From the VIEW TYPE menu, choose **General**, **Full**, **Section**, **No Scale** (or **Scale**), and **Done**.
3. Do one of the following:
   - To create an *unfolded* cross-sectional view, choose **Full**, **Unfold Xsec**, and **Done** from the XSEC TYPE menu.
   - To create a *total unfolded* cross-sectional view, choose **Full**, **Total Unfold**, and **Done** from the XSEC TYPE menu.

   Note: Unfold Xsec and **Total Unfold** are *only* available if you choose **Full** from the XSEC TYPE menu.
4. Select the location for the view.
5. Select or create a section using commands in the XSEC ENTER menu.
6. Select the view for showing arrows and set the direction of viewing by choosing DIRECTION > **Flip** > **Okay**. If you do not select a view to show arrows, press the middle mouse button.
7. After the view appears, you can leave it as is or orient it by rotating it at an angle. Choose FLAT ORIENT > **Angles** > **Norm**. Type the value for the rotation angle (an angle about the normal axis), and choose **Done/Accept**. The view appears in its new orientation.
8. To conclude view creation, choose **Done Orient**. To abort the process, choose **Quit Orient**.

About Location Callouts

You can define a zone or location grid to drawing formats by creating location callouts. You define the location grid by using the existing lines of a format or by using equally spaced increments. Use this location grid to locate drawing views using parametric notes to indicate both the sheet and drawing location of a view. Parametric location callouts allow for intelligent notes to be added to drawings to indicate the location of a detail or section view, saving time and preventing out-of-date notes.

Location callouts are created using a standard syntax:

- View location—`&pos_loc:view:<view name>`
- Sheet of view—`&sheet:view:<view name>`
- Note location in parent view—`&pos_loc:parent_note:<view name>`
- Sheet location of note—`&sheet:parent_note:<view name>`

For example, if Detail A is located in A3 on the location callout grid and on sheet 2, type the note:
`See Detail A at &pos_loc:view:detail_a on sheet &sheet:view:detail_a`

The note text shows as:

See Detail A at A3 on sheet 2.

The syntax for the location callouts is customizable, which allows for additional text to be added based on individual company standards. The detail setup option `pos_loc_format` is provided to specify the syntax of the `pos_loc` and sheet parameters in the callout. The default of `pos_loc_format` is %s%x%y, and appears as 2D4.

%r added to `pos_loc_format` will repeat the syntax string for the same item appearing several times in different locations. This applies to items such as symbols and connectors, but not to views because views have unique names. The syntax will be %s%x%y, %r where %r means to repeat the substring from the first special character to the last.

`&pos_loc` Behavior:

The location note updates upon regeneration if the view is moved on the drawing. If the view is erased or moved outside of the grid, the value of the location callout changes to ***. The callout updates when the view is resumed or the view is moved back onto the grid. If the view is deleted, then the location callout is no longer parametric and the value changes to ***.

`&sheet` Behavior:

If you switch the view to a different sheet, the location callout automatically updates to the new sheet number. If the view is erased, the value of the location callout changes to ***. The callout updates when the view is resumed. If the view is moved outside of the grid, the value remains as the correct sheet number. If the view is deleted, then the location callout is no longer parametric and the value changes to ***.

You specify which corner of the location grid you want as the origin. The grid origin is shown as a yellow X. The default grid origin is the lower right corner of the location grid.

To Define the Location Callout Grid

1. In Format mode, click **Advanced > Location Grid > Define**.
2. Choose one of the following to define:
   - **Grid Outline**—Specify the grid outline by using the rubberband box or by papersize (equal to the size of the page).
   - **Grid Origin**—Pick a point to indicate the grid origin. The lower right-hand corner is the default origin.
   - **Columns**—Specify the amount and location of grid columns by number or pick points, and then specify the grid labels by letter, number, or custom.
   - **Rows**— Specify the amount and location of grid rows by number or pick points, and then specify the grid labels by letter, number, or custom.
3. Click **Done/Return**.

To Show the Location Callout in a New Drawing

1. In Drawing mode, click **Insert > Note**.
2. Select the location for the location callout.
3. Type a note, and then type the following:
   - To show the view location on the grid: `&pos_loc:view:<view name>`
   - To show just the sheet location: `&sheet:view:<view name>`
   - To show the note location in the parent view: `&pos_loc:parent_note:<view name>`
   - To show the sheet location of the note: `&sheet:parent_note:<view name>`
4. Click **Done/Return**. The location callout is created.

Example: A Location Callout

To show the location of a view named Main_view, type the following in the **Enter NOTE** text box:
```
Main_view is located in &pos_loc:view:main_view
```

The note shows as:

```
Main_view is located in 3B
```

According to the location callout grid you specified, Main_view is located in section 3B.

About Manipulating Views

Using commands in the VIEWS menu, you can manipulate views in several ways:

- Move views
- Switch views to another sheet
- Delete views
- Relate views to draft entities
- Erase and resume views

Moving Views

If you move a view from which other views were projected (parent view), the projected views (children) also move to maintain view alignment. For example, if you move the front view in the following figure vertically, the right side view also moves to maintain alignment because it was projected from the front view.

You can only move the front view vertically because it is a child of the top view and must maintain alignment with it. This alignment and parent/child relationship between projected views remains intact even as the model changes. You can move general and detailed views to any new location because they are not projections of other views.

Using the GET POINT menu, you can do exact drafting to place the view where you want it. For example, to exactly align one view with another general view, set the view origin using **Origin** in the MODIFY VIEW menu and the GET POINT menu. This establishes a reference point for moving the view, so that you can easily place it anywhere on a drawing relative to another view.

To Move a View

You can move the origin of a view or any point on the view to a new location.

1. Select the view to move and then click the right mouse button.
2. Click **Move**. The borders of all views appear, along with handles at the four corners of the border and at the origin.
3. You can move the view to any location using a drag-and-drop operation.
   To move the origin or a corner of a view, select a point close to the view's handle. To move another point of the view, select a point on the view. To do exact drafting, use the GET POINT menu commands (such as **Rel Coords** and **Abs Coords**) to specify the exact coordinates, an offset, entity, or vertex.
4. Select the desired location for the view. The view moves to the selected location.

To Switch Views to Another Sheet

You can move drawing views from one sheet of a drawing to another using the **Switch Sheet** command in the SHEETS menu.

To Delete a View

To delete views in a drawing, use the **Delete View** command in the VIEWS menu. You cannot remove a parent view while its children still exist; you must first remove the child views.

Pro/DETAIL

To Relate Detail Items to a View

Using the **Relate View** command in the **VIEWS** menu, you can associate detail items with a specified view in a multimodel drawing.

About Erasing and Resuming Views

You can erase a drawing view from a drawing without affecting other views, notes, or cross-section arrows. Erasing and resuming views is an effective technique for improving the view regeneration and repaint times of complicated drawings.

The following rules apply:

- If a note or symbol is attached to the erased view as well as to other views, the system also erases the leaders attached to the erased view. When you resume the view, the leaders reappear.

- You cannot show dimensions on other views if they appear on the erased view.

- If you erase a parent view of a detailed view containing a local area cross section, the system transforms the local cross section in the detailed view into a full cross section.

- When you resume a view that was erased from the current sheet, the name of the view appears in the menu for selection, and the outline of the view appears on the sheet.

- When you resume a view that was erased from another sheet, the name of the view appears in the menu for selection. No outline appears. You can still resume the view to a different sheet.

- To control the display of erased view outlines use the environment option **Highlight Erased Views**. You can control this option using the configuration file option `highlight_erased_dwg_views` (the default is `yes`).

- Erased view outlines and names are not plotted.

To Erase a View

1. Select a view to erase.
2. Remove any arrows or circles associated with the view (if they are present).
3. Choose **VIEWS > Erase View**. The system erases the view.

To Resume an Erased View (or Views)

1. Click **VIEWS > Resume View**.
2. Select view names from the namelist menu. If you want to apply the change to all views, choose **Select All**. To unselect all of the views, choose **Unsel All**.
3. A cyan border outlines the selected views. Choose **Done Sel** and the views reappear.
   Note: If you switch the view to another sheet and then erase it, you can resume it on the original sheet or on the current one.

About Using Model Colors in Drawings

You can toggle color display of selected views in Drawing mode between the assigned drawing colors and the colors used in the original model. Click *See Also* for more information.

Important Points about Using Model Colors in Drawings

- Views that are displayed in the original model colors are plotted (printed) as such.

- Changing the color of individual geometry does not change the colors in views that display the original model colors.

- If you change model colors in the associated model, the new model colors are updated automatically in the drawing, if you set the drawing views to use model colors.

- Hidden lines in drawing views always take on the original standard hidden line color in the model.

- Any assigned process assembly colors always supercede the setting for using model colors or assigned drawing colors in the drawing.

To Use Model Colors in Drawing Views

1. On the DRAWING menu (Menu Manager), click **Views > Disp Mode > View Disp**. The GET SELECT menu appears.
2. Select a view or views for which you want to use the original model colors, and then click **Done Sel**.
3. In the VIEW DISP menu, click **Model Color**, and then click **Done**. The model colors are displayed in the selected views.

Note: Model colors and assigned drawing colors in a drawing for a process assembly are always superceded by any existing process assembly colors.

To Use Drawing Colors in Drawing Views

1. On the DRAWING menu (Menu Manager), click **> Views > Disp Mode > View Disp**. The GET SELECT menu appears.
2. Select a view or views for which you want to use the assigned drawing colors, and then click **Done Sel**.
3. In the VIEW DISP menu, click **Drawing Color**, and then click **Done**. The drawing colors are displayed in the selected views.

Note: Model colors and assigned drawing colors in a drawing for a process assembly are always superceded by any existing process assembly colors.

About Modifying Crosshatch

Using the **Edit > Properties** command, you can modify the crosshatch of detailed views and individual members of assembly cross-sectional views.

- *Assembly cross sections*—When modifying an assembly cross section, you can modify individual components of the assembly intersected by the cross section.
- *Detailed views*—The crosshatch of a detailed view can follow that of its parent view, or you can make the detailed view independent of its parent. To modify the crosshatch of a detailed view, you must make the detailed view independent of its parent by using the **Det Indep** command in the MOD XHATCH menu.

  Note: If you have overlapping crosshatch patterns in your drawing (any combination of crosshatched cosmetic features, cross-sectional views, or draft cross sections), choose **Query Sel** from the GET SELECT menu to select the crosshatch to modify.

Using Smart Default Crosshatch

When you create an assembly cross section, the system automatically applies a "smart" crosshatch pattern on the parts that provides a superior visual representation of the cross section. Smart crosshatch applies crosshatch spacing appropriate to the model size and assigns different angles to different parts in the assembly. Smart crosshatch uses a randomized slant angle between adjacent components, making it easier to distinguish different parts in assembly drawings and also reducing the amount of time required to clean up drawings with cross sections and crosshatch.

The following rules apply:

- Smart default crosshatch affects newly created cross-sectional views only. When you retrieve previously saved drawings, smart crosshatch is not in effect.
- For assembly cross sections, smart default crosshatch applies to both spacing and angle. For part cross sections, smart default crosshatch applies to spacing only. By default, the angle is 45 degrees for parts.
- To override smart default crosshatch for newly created cross sections, define cross section parameters `default_xhatch_spacing` and `default_xhatch_angle`. Cross sections of other surrounding parts that are not defined adjust accordingly.
- Smart default crosshatch no longer applies after you modify the crosshatch until you create a new view.

- Modifications made in Drawing mode do *not* display in Part mode or Assembly mode. Modifications automatically override smart default crosshatch in a drawing.

Controlling the Spacing and Angle of Crosshatch Using Parameters

By setting parameters, you can control the default spacing and angle of crosshatch in newly created planar and offset cross sections. To control the display, set the number parameters `default_xhatch_spacing` and `default_xhatch_angle` using the SETUP menu or the RELATIONS menu. These parameters only affect *new* cross sections—they do *not* affect cross sections previously created in a part or assembly.

Modifying Crosshatch Characteristics

You can modify the following characteristics of crosshatch, as well as create a filled area and retrieve and save crosshatch patterns:

- Spacing
- Angle
- Offset
- Line style
- Number of lines

To Modify Crosshatch Patterns

1. Select the cross-sectional views to modify.
2. Click **Edit > Properties**..
3. Using the **MOD XHATCH** menu on the Menu Manager , do one of the following:
   - Modify the spacing. Choose **Spacing**; then choose **Half**, **Double**, or **Value** from the MODIFY MODE menu. Specify which lines to change by choosing **Individual** or **Overall**. The system automatically scales the crosshatch offset, as well as the spacing, according to the command you choose for spacing.
   - Modify the angle. Choose **Angle**; then choose **Individual** or **Overall** from the MODIFY MODE menu. The system changes only the first line of the pattern. Choose an angle (**0**, **30, 45**, **60**, **90**, **120**, **135**, **150**) or **Value** to specify a different angle.
   - Modify the offset. Choose **Offset**; then type a value for the offset in drawing units.
   - Modify the line style. Choose **Line Style**; then use the Line Style dialog box to change the line style.
   - Modify lines in the pattern. Choose **Next Line** and **Prev Line** to select a pattern line; then choose **Add Line** or **Delete Line**.

   Note: You can also modify crosshatch characteristics to cosmetic features and closed datum curves.

About Creating a Filled Area

You can display cross-sectional drawing views as filled areas (that is, displayed in solid color) by changing the crosshatch to fill, but you must first create a cross-sectional view. To switch between a hatched and filled cross section, use the **Hatch** and **Fill** commands in the MOD XHATCH menu.

When creating filled cross-sectional drawing views, keep in mind the following rules:

- You can create filled cross sections only in views without clipping. If you attempt to change the crosshatch of a clipped view (such as a detailed view, broken view), the system issues an error message and does not allow you to proceed.
- If a cross-sectional view that you want to fill has detailed views, the system warns you that this view has dependent children views before allowing you to make the change. The detailed view appears without either crosshatch or filling; you can then set its crosshatch to be independent of its parent.
- You can modify only planar cross sections to be filled.

To Create a Filled Cross-Sectional View

1. Select the cross section.
2. Choose **Edit > Properties > MOD XHATCH > Fill** > **Done**. .
3. Once you have defined the cross section as filled, you can use **Line Style** to set the appropriate color for the filled area, or **Save** to save it to disk with a ".xch" extension. Later, you can use **Retrieve** to read it in as a saved crosshatch pattern.

Saving Crosshatch Patterns

You can assign a name to a crosshatch pattern and save it for later retrieval by using the configuration file option `pro_crosshatch_dir` to specify a default directory for crosshatch patterns. Its value is the full path name of the default directory.

To Save a Crosshatch Pattern

1. Select the view containing the crosshatch pattern that you want to save. For an assembly cross section, you may need to use **Next Xsec** or **Prev Xsec** in the MOD XHATCH menu to select the member.

2. Click **Edit > Properties**.

3. On the Menu Manager, click **Save**.

4. Type a name for the crosshatched pattern.

5. If you have not set the configuration file option `pro_crosshatch_dir`, the system saves the crosshatch pattern in the current directory, giving it the name that you specified and the extension `.xch`. If you want to save the pattern in a separate directory containing crosshatch patterns, you must save the pattern first, and then move it to its new location using operating system commands.

To Retrieve a Crosshatch Pattern

1. Select a cross-sectional view.
2. Click **Edit > Properties**.
3. Choose MOD XHATCH > **Retrieve**.
   Pro/ENGINEER searches through the current directory, the `pro_crosshatch_dir` configuration file option, and the system directory. It then displays a namelist menu that contains the names of any crosshatch patterns in the current directory, all crosshatch patterns in the default crosshatch directory, and the nine standard crosshatch patterns provided with Pro/DETAIL.
   Note: The selected crosshatch pattern replaces the view's (or assembly member's) current crosshatch. When you change the crosshatch type in the drawing, the model does not reflect it; conversely, if you change it in a model, its drawings do not reflect it.

Examples: Crosshatch Patterns
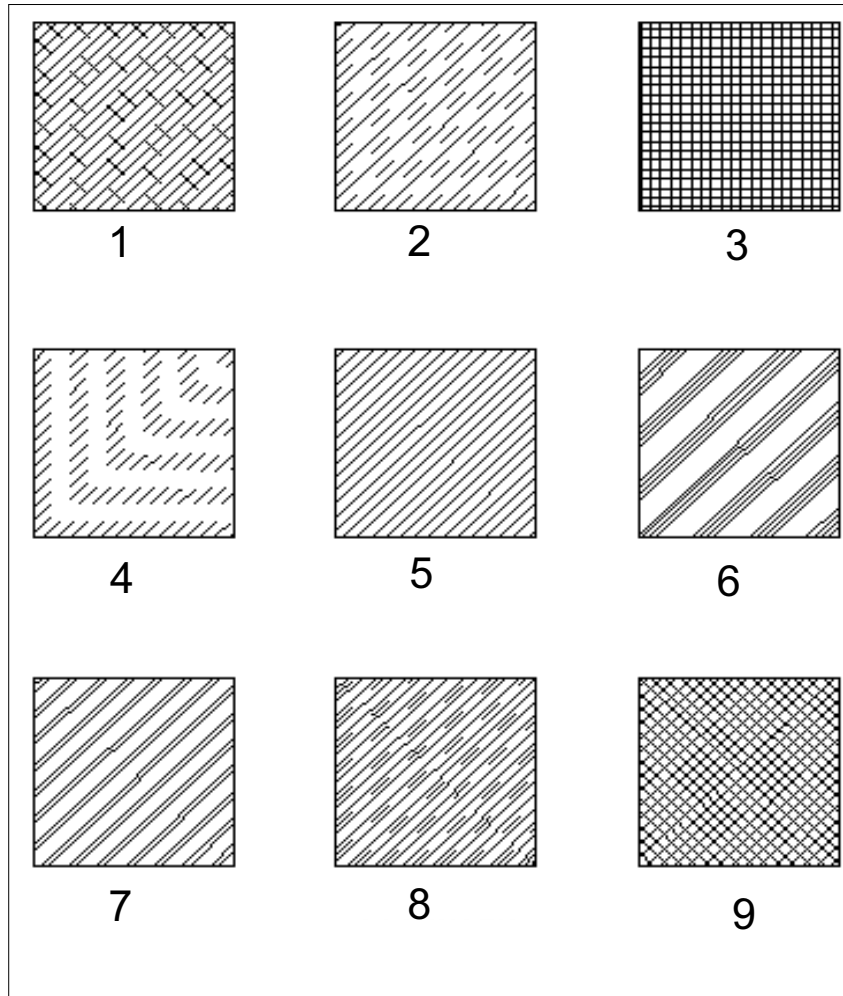
1   Aluminum.
2   Copper.
3   Electric.
4   Glass.
5   Iron.
6   Plastic.
7   Steel.
8   Titanium.
9   Zinc.

Modifying Assembly Cross Sections

When modifying an assembly cross section, the MOD XHATCH menu displays five additional commands: **Excl Comp**, **Restore Comp**, **Next Xsec**, **Prev Xsec**, and **Pick Xsec**.

- Using **Excl Comp** and **Restore Comp**, respectively, you can exclude components of an assembly cross section from or resume them in the cross-sectional view.

- Using **Next Xsec** and **Prev Xsec**, you can navigate through all components.

- Using **Pick Xsec**, you can select directly in the graphics window instead of navigating through all components (this command is also available when you are modifying crosshatch in Assembly mode).

To Modify the Display of an Assembly Member in a Cross Section

1. Select the cross-sectional view to modify.
2. Select the appropriate member of the assembly cross section.
3. Click Edit > Properties.
4. Choose **Excl Comp** on the Menu Manager to remove a member from the cross-sectional display, or choose **Restore Comp** to resume a member that you have excluded. Surfaces of a member that you have excluded are displayed as they would be in a noncross-sectional view, unless the excluded member is part of an area cross section. If you exclude a member of an area cross section, it disappears entirely.
   Note: Crosshatch is not affected by HLR and is therefore not obscured by excluded components.

About Modifying Display Mode

Using the **Disp Mode** command in the VIEWS menu, you can change the display mode (**Hidden Line**, **Wireframe**, **No Hidden**) of an individual view, edge, or assembly member.

The configuration file option `hlr_for_quilts` controls how the system displays quilts in the hidden line removal process. If you set it to `yes`, the system includes quilts in the hidden line removal process. If you set it to `no`, the system does not include quilts in the hidden line removal process.

To Modify the Display Mode of Individual Views

1. Choose **VIEWS > Disp Mode > View Disp**.
2. Select views to modify; then choose **Done Sel**.
3. From the VIEW DISP menu, choose one of the following:
   - **Wireframe**—Sets the display mode to wireframe
   - **Hidden Line**—Sets the display mode to hidden line
   - **No Hidden**—Sets the display mode to no hidden
   - **Default**—Sets the display mode as set in the Environment dialog box
   - **Qlt HLR**—Includes quilts in the hidden line removal process (except cross-sectional views)
   - **No Qlt HLR**—Excludes quilts from the hidden line removal process in all views (except cross-sectional views)
   - **Tan Solid**—Displays tangent edges
   - **No Disp Tan**—Turns off the display of tangent edges
   - **Tan Ctrln**—Displays tangent edges in centerline font
   - **Tan Phantom**—Displays tangent edges in phantom font
   - **Tan Dimmed**—Displays tangent edges in dimmed color
   - **Tan Default**—Sets the display of tangent edges as set in the Environment dialog box
4. Choose **Done**. The system updates the selected views.
   Note: Once you have set the display mode for a specific view, it remains set regardless of the setting in the Environment dialog box, unless you choose **Default** in the VIEW DISP menu.

Disallowing the Selection of No Hidden Edges in Drawings

With complex models, it is sometimes difficult to find the exact edge you want to select. When you are in No Hidden display mode, you can disallow selection of edges in drawings while using Query Select.
The configuration file option `select_hidden_edges_in_dwg` enables you to disallow the selection of

No Hidden edges in drawings. This configuration file option disallows the selection of No Hidden edges by rejecting edges behind the first surface in the viewing plane. The following requirements apply:

- Selection is disallowed for drawing views in No Hidden display mode only.

- View regeneration is required on cross-sectional views.

- You can disallow the selection of edges in general and cross-section views.

To Modify the Display of a Detailed View

After making a detailed view independent of its parent, you can specify the circle representation of a detailed view, as well as change the type of entity around the parent view.

1. Click **Modify View > DISP MODE**. The **DISP MODE** menu appears.
2. Click **View Disp**.
3. Select a detailed view on the screen; then choose **Done Sel**.
4. Click **View Disp > Det Indep**.
5. Choose the appropriate specifications and then click **Done**.

To Specify the Circle Representation of a Detailed View

1. Click **VIEWS > ADD VIEW > Detailed** and then click **Done**.
2. Select a center point location for the drawing view.
3. Type a scale dimension for the view and then click .
4. Select the center point for detail on an existing view.
5. Type a name for the detailed view.
6. Sketch the spline around the area that you want to detail.
7. Click one of the following from the **BOUNDARY TYPE** menu:

   - **Circle**—Draws a circle in the parent view for the detailed view.

   - **Ellipse**—Draws an ellipse in the parent view for the detailed view to closely match the spline, and prompts you to select an attachment point on the ellipse for the view note.

   - **H/V Ellipse**—Draws an ellipse with a horizontal or vertical major axis and prompts you to select an attachment point on the ellipse for the view note.

   - **Spline**—Displays the actual spline boundary on the parent view for the detailed view, and prompts you to select an attachment point on the spline for the view note.

   - **ASME 94 Circ**—Displays an ASME standard compliant circle in the parent view as an arc with arrows and the detailed view name.

8. Except for **ASME 94 Circ**, choose a location for the detailed view note. The note in the parent view is attached to the chosen boundary entity and places another detailed note below the detailed view.

9. The parent view appears with the specified boundary. You can modify the detailed view boundary and reference point on the parent view by clicking **VIEWS > Modify View > Boundary** and selecting the detailed view.

Note: You can convert detail view boundary types to an ASME 94 circle. However, once a view is displayed in ASME94, it cannot be changed.

To Change the Type of Entity Around the Parent View

1. Click **VIEWS > Modify View > Boundary**.
2. Select a detailed view.
3. From the VIEW BNDRY menu, choose one of the following:
   - **Circle**—Draws a circle in the parent view for the detailed view. Creates the detailed note in the parent

view attached to the chosen boundary entity and places another detailed note below the detailed view.

- – **Ellipse**—Draws an ellipse in the parent view for the detailed view to closely match the spline, and prompts you to select an attachment point on the ellipse for the view note.
- – **H/V Ellipse**—Draws an ellipse with a horizontal or vertical major axis and prompts you to select an attachment point on the ellipse for the view note.
- – **Spline**—Displays the actual spline boundary on the parent view for the detailed view, and prompts you to select an attachment point on the spline for the view note.

The system modifies the entity around the parent view as specified.

To Change the Edge Display of Detailed Views

If the edge display of a detailed view is the same as that of its parent view, you cannot use the **Edge Disp** command in the DISP MODE menu to modify it independently of the parent. To change the edge display of a detailed view, choose **Edge Disp** and select the *parent* of the view. You must choose **Regen View** from the Pro/ENGINEER **View** menu (*not* the **Repaint** command) to update the display of the detailed view properly after modifying the parent view.

Note: You can change *only* the edge display of the detailed view if it is different from that of its parent view.

Manipulating Edge Display

Using commands in the VIEWS > DISP MODE > Edge Disp menu, you can set the display mode for individual edges that are usually visible on the screen. To modify edge display, clear the **Wireframe** command in the Environment dialog box. *All model edges must be visible.*

You should use the **Edge Disp** command in the DISP MODE menu to make minor cosmetic changes, and to unblank selected hidden edges in views with the display mode set to **No Hidden**. You cannot use it to erase the following items:

- Boundaries of all cross sections
- Outside edges intersecting clipped splines
- Cosmetic threads in drawing views (choose **Show and Erase** from the View menu; then click the **Cosmetic Feature** button in the **Type** box of the Show/Erase dialog box)
- Silhouette edges, except for the following:
  - – Silhouette edges of cylinders and cones
  - – Silhouette edges of torii that are arcs, that is, the axes of the torii are either parallel or perpendicular to the screen
  - – Silhouettes of general surfaces of revolution whose axes are parallel to the screen

  Note: If you erase one silhouette edge, the system erases all silhouette edges that belong to the same surface as well.

Selecting Entities

When the configuration file option "selection_of_removed_entities" is set to "yes," you can select entities and activate them for performing such actions as showing dimensions and placing items on a layer. You can also activate entities for choosing features that are in front of a cross section (planar or offset), have been removed via Z-Clipping, or have been erased using **Erase Line** in the EDGE DISP menu.

Note: If you have set the view display mode to **Wireframe**, the system does not erase or redisplay its edges until you change the view display to **Hidden** or **No Hidden**.

Tangent Lines

Using commands in the TAN EDGEDISP menu (a submenu of the EDGE DISP menu), you can select certain tangent lines to display in drawing views, while erasing others:

- **Tan Solid**—Displays selected tangent edges regardless of the tangent edge display setting in the Environment dialog box. A tangent edge is the intersection at which two surfaces become tangent. For

example, a round always produces a tangent edge with the surfaces it intersects.

- **Tan Ctrln**—Displays selected tangent lines in the centerline style regardless of the tangent edge display setting in the Environment dialog box.
- **Tan Phantom**—Displays selected tangent lines in the phantom line style regardless of the tangent edge display setting in the Environment dialog box.
- **Tan Dimmed**—Displays selected tangent edges in dimmed color.
- **Tan Default**—Displays tangent edges according to the setting in the Environment dialog box.

  Note: The **Erase Line** and **No Hidden** commands in the EDGE DISP menu take precedence over the TAN EDGEDISP menu commands. If you choose **Erase Line** or **No Hidden**, the selected tangent edge does not appear.

If you set the drawing setup file option `hidden_tangent_edges` to `dimmed`, you can plot hidden tangent edges in a view using Pen 7. The lines appear dashed in the same color as the dimmed visible tangent edges. However, you must select **Hidden Line** or **No Hidden Line** from the **Display Style** list in the Environment dialog box. If you set this option to `erased`, the system removes all hidden tangent edges automatically from the screen and the plot.

To Modify Display of Individual Edges

1. Choose **VIEWS > Disp Mode > Edge Disp**.
2. Choose one of these commands from the EDGE DISP menu:
   - **Erase Line**—Erases a visible line from the view display
   - **Wireframe**—Shows a selected edge in wireframe style
   - **Hidden Style**—Shows a selected edge in hidden line style
   - **Hidden Line**—Shows a hidden edge as a hidden line
   - **No Hidden**—Removes a hidden edge from view display
   - **Default**—Displays the edge using the current environment setting
3. Specify the selection technique by choosing one of these commands from the EDGE DISP menu:
4. Choose GET SELECT > **Pick Many** to select the edges. The system selects coincident edges covered entirely by the top edge.
5. Choose ERASE DISP > **Highlight Mod** to highlight all modified edges of the model.

Examples: Types of Edge Displays
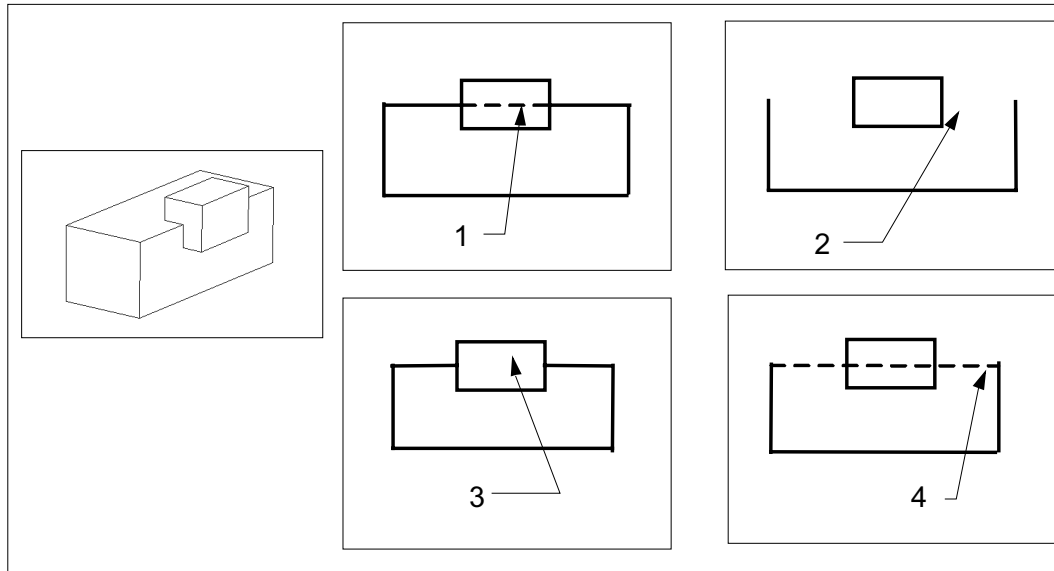
1   Hidden.
2   Erase.
3   No Hidden.
4   Hidden Style

Manipulating the Display of Assembly Members

You can manipulate the display of assembly members by changing their hidden line display, changing their line style, or blanking and unblanking them.

If you choose **Member Disp** from the DISP MODE menu, the MEMB DISP menu displays the following commands:

- **HLR Display**—Changes hidden line display of selected components, overriding the view display.
- **Style**—Modifies the line style of selected assembly members.
- **Blank**—Removes selected assembly members from the display.
- **Unblank**—Returns previously blanked components to the display. Temporarily outlines in blue only the views containing blanked components. Select the appropriate view in order to highlight blanked components for selection. You can also select components from the Model Tree.

You can also specify where you want to effect the changes by choosing **Picked View**, **This Sheet**, or **All Views** from the MEMB DISP menu.

Changing the Hidden Line Display

You can change the hidden line display of selected components (overriding the view display), using the **HLR Display** command in the MEMB DISP menu. You can set the display of selected components on a per view or per sheet basis, or in all views.

If you choose **Hidden Line** from the HLR DISPLAY menu, and then select a component, the results are as follows:

- If you have set the view to **No Hidden**, the system shows the component's hidden lines.
- If you have set the view to **Hidden Line**, it does not affect the component's hidden line display.
- If you have set the view to **Wireframe**, it does not affect the component's hidden line display.

If you choose **No Hidden** from the HLR DISPLAY menu, and then select a component, the results are as follows:

- If you have set the view to **No Hidden**, it does not affect the component's hidden line display.
- If you have set the view to **Hidden Line**, the system does not display the component's hidden lines.
- If you have set the view to **Wireframe**, it does not affect the component's hidden line display.

If you have selected one component, or all of the components that you have selected have the same setting, the HLR DISPLAY menu indicates the setting by highlighting the command in the menu. If you have selected more than one component, but they have different settings, the system does not highlight a command in the HLR DISPLAY menu.

> **Note:** The setting assigned by the **Edge Disp** command takes precedence over any setting assigned by the **HLR Display** command, but the setting assigned by the **HLR Display** command takes precedence over any setting assigned by the **View Disp** command.

Display of Pipe Centerlines

If you set the drawing setup file option `hlr_for_pipe_solid_cl` to `yes`, hidden line removal affects pipe centerlines. If you set it to `no` (the default), it does not.

Note: The `hlr_for_pipe_solid_cl` option operates only on pipes created in Pro/PIPING, not on pipe features in a part.

To Modify the Line Style of Selected Assembly Members

1. Click **VIEWS > Disp Mode > Member Disp > Style**.
2. Select the assembly components; then choose **Done Sel**. You can use **Sel By Menu** in the GET SELECT menu to select parts or subassemblies for modification.
3. Change the style by selecting a command from the MEMB STYLE menu:
   - **Standard**—Displays the selected member view in solid line style.
   - **PhantomOpque**—Displays the selected member view in phantom line style.
   - **PhantomTrnsp**—Displays the selected member view in phantom line style, but the hidden line removal process does not affect it.
   - **User Color**—Creates a user-defined color and assigns it to an assembly member.
4. Choose **Done**.

Examples: Modified Line Styles of Selected Assembly Members

If you set the view using **No Hidden**, the system hides the objects behind it if you display them in the **PhantomTrnsp**, **PhantomOpque**, or **Standard** style.

Examples Using Standard Font
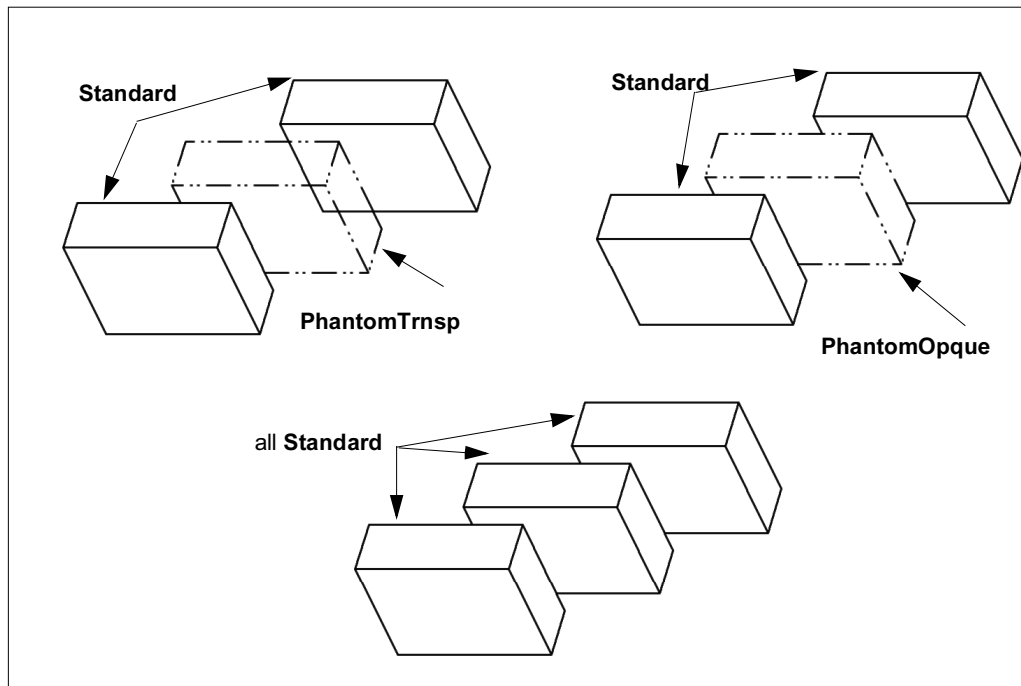
If you change the line style of a selected member to **PhantomOpque**, it appears in phantom line font. If you set the view using **No Hidden**, the system hides the objects behind it if you display them in the **PhantomOpque**, **PhantomTrnsp**, or **Standard** style.

Examples Using PhantomOpque Style
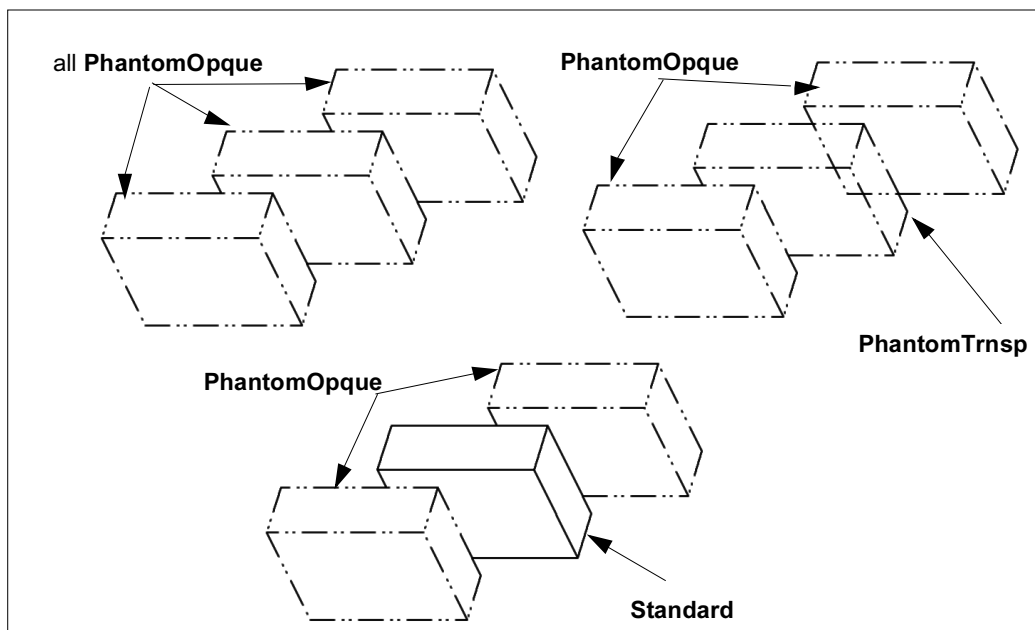
If you change the line style of a selected member to **PhantomTrnsp**, it appears in phantom line font. If you set the view using **No Hidden**, the system hides the objects behind it if you also display them in the **PhantomTrnsp** style. However, if you display the objects in the **PhantomOpque** or **Standard** style, it does not affect the hidden line removal process, so you can see their edges.

Examples Using PhantomTrnsp Style
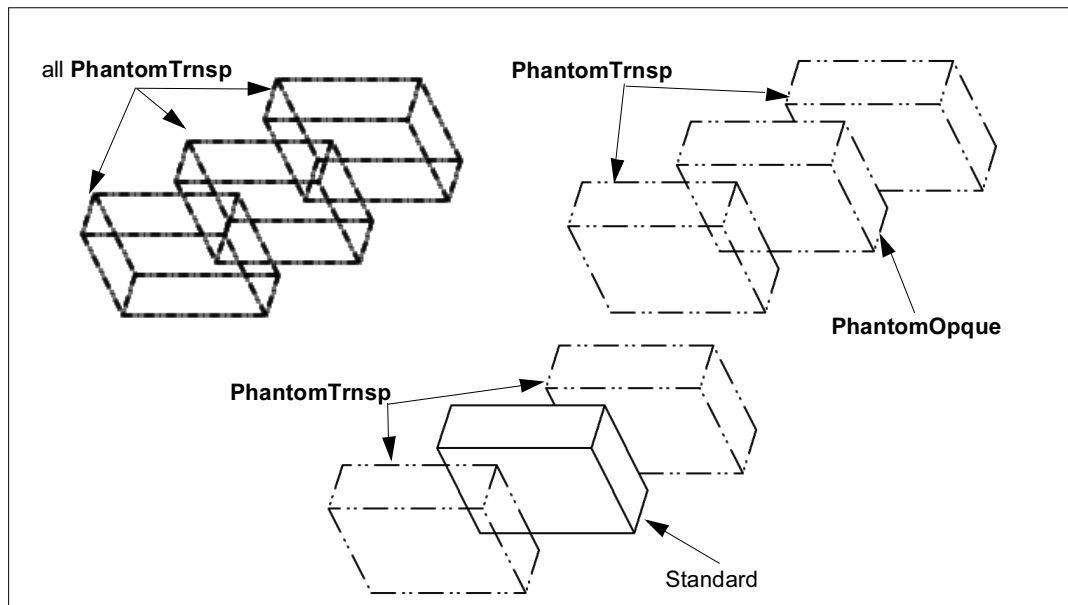
About Drafting in Drawing Mode

In Draft mode, you can do the following:

- Create construction geometry
- Chain geometry
- Make dimensions of draft entities associative
- Create draft geometry
- Define draft cross sections
- Modify, copy, and move draft geometry
- Obtain information about draft geometry

You can add draft entities to a drawing at any time. Pro/ENGINEER considers the following to be draft entities:

- Entities created using the Skecth menu (for example, 2-D draft geometry).
- Entities selected during the creation of the entity (for example, attaching a geometric tolerance symbol to a draft dimension).
- Entities read in to a drawing from an IGES, DXF, or SET file.
- Annotations placed without any associativity to a Pro/ENGINEER model (for example, a free note).

Draft Geometry

Using commands in the Sketch menu, you can create various geometry types: lines, circles, arcs, splines, ellipses, points, and chamfers.

To Create a Construction Line

Construction geometry entities are lines and circles that you can use to locate and create 2-D draft geometry. They appear on the screen in phantom font, but you can transfer them through IGES files and plot them.

1. Click **Sketch > Construction Line**.
2. Using the **Construction Line** menu, do one of the following:
   - Choose **Horiz** or **Vert** to create a horizontal or vertical line. Use commands in the GET POINT menu to locate the lines.

- Choose **Crossed Pair** to create two construction lines perpendicular to each other at a given angle. Use the GET POINT menu to select a point on the drawing; then type a rotation angle. The system creates two construction lines, with both lines passing through the selected point. One of the lines is at the specified angle; the other is perpendicular to the first.
- Choose **Angle** to create an angled line. Use the GET POINT menu commands to locate the pivot point of the line; then type an angle for every line that you want to create. Each angle is absolute—a positive value for counterclockwise and a negative value for clockwise. Press ENTER on an empty line to quit.
- Choose **Offset** to create a line offset from another. Select the line from which to offset; then type an offset value. A positive value offsets in the direction of the arrow; a negative value reverses the direction. Each offset that you type after the first one is relative to the line you just created. Press ENTER on an empty line to quit.
- Choose **Pnt/Angle** to create a construction line through a point and at a specified angle to a selected entity. Use the GET POINT menu to select a point that the construction line will pass through. Then select an entity that you want as the angular reference. Type an angle between –360° and 360°. The construction line at the specified angle is created counterclockwise from the selected entity.

Tip: Creating a Construction Line

Note: You *cannot* offset a construction line from an axis line.

Example: Construction Lines
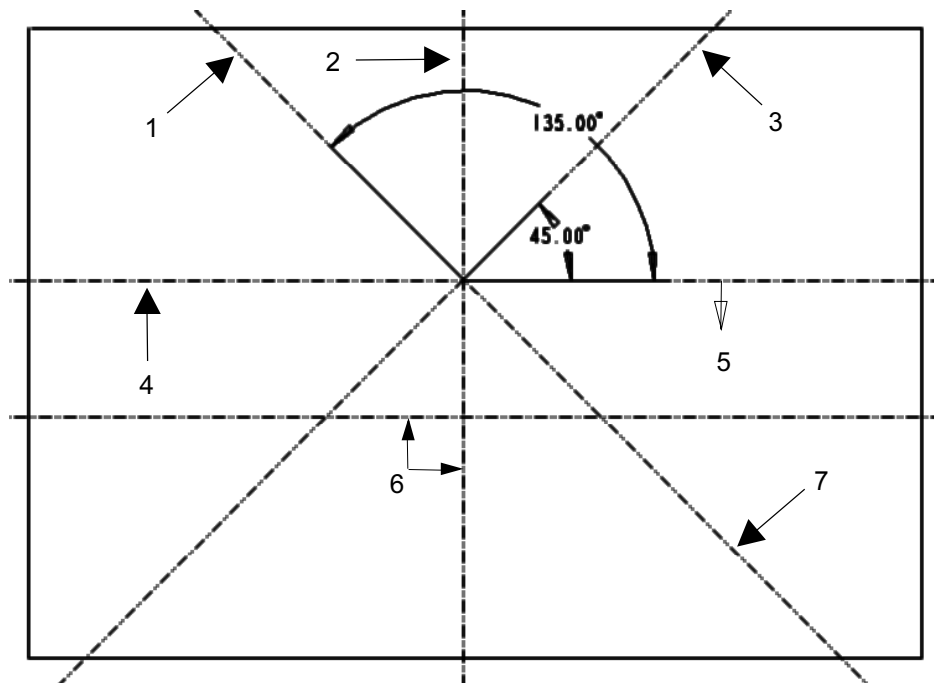
1	Angle line.
2	Vertical line.
3	Angle line.
4	Horizontal line.
5	Offset direction arrow.
6	Perpendicular lines.
7	Offset line.

To Create a Construction Circle

1. Click **Sketch > Construction Circle**.
2. Use commands in the CIRCLE menu to create a circle.

Example: Construction Circles
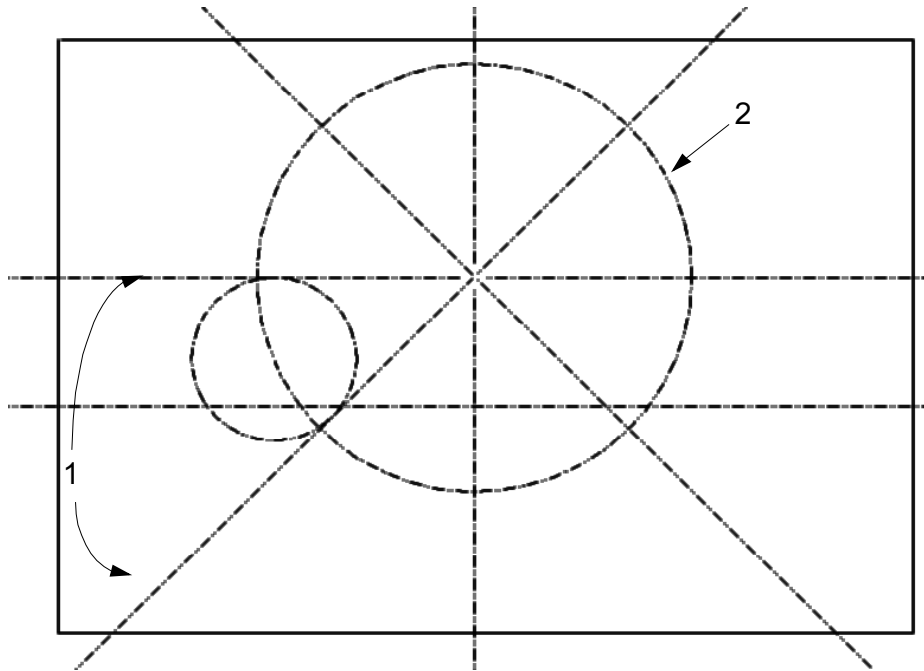
1   This circle is filleted to two lines.
2   This circle is centered in the drawing and passes through the center of the other circle.

To Chain Entities During Sketching

You can chain geometry to create a series of entities by selecting points on previous entities as starting points for new ones.

To initiate chaining, click **Sketch > Chain**. To end chaining, choose **End Chain**, or exit the **DRAFT GEOM** menu. When chaining is in effect, a small yellow square indicates the point from which the chain continues. When sketching entities while chaining is in effect, if you select points with the left mouse button, the chain continues; if you select them with the middle mouse button, the system creates an endpoint and the chain *pauses*, or stops at that point. If you have finished creating chained draft entities, you can choose **End Chain** or continue making draft geometry. The newest entity begins where you paused at the end of the last chain. To begin a different set of chained entities, choose **End Chain** to finish the first chain and choose **Start Chain** to start a new one; otherwise, the new chain of draft entities simply continues from the previous one.

Chaining Entities During Sketching

Chaining geometry affects only the creation of the entities. Once you have created them, each entity exists independently of any others, regardless of any shared center points or endpoints.

You can chain together circles and ellipses when they use the same centers. Once you have established the center of the first ellipse or circle of the chain, the system uses it for every circle or ellipse that follows, until you end the chain.

Examples: Chaining Geometry
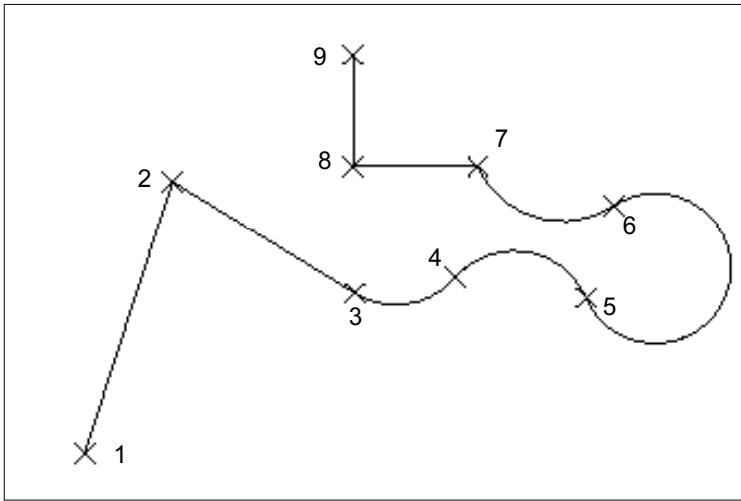
Chaining geometry using endpoints.

1   Start. Choose **Line** and select first point.
2   Pick 2.
3   Pick 3. Then select **Arc** and continue.
4   Pick 4.
5   Pick 5.
6   Pick 6.
7   Pick 7. Then select **Line** and continue.
8   Pick 8.
9   Pick 9.
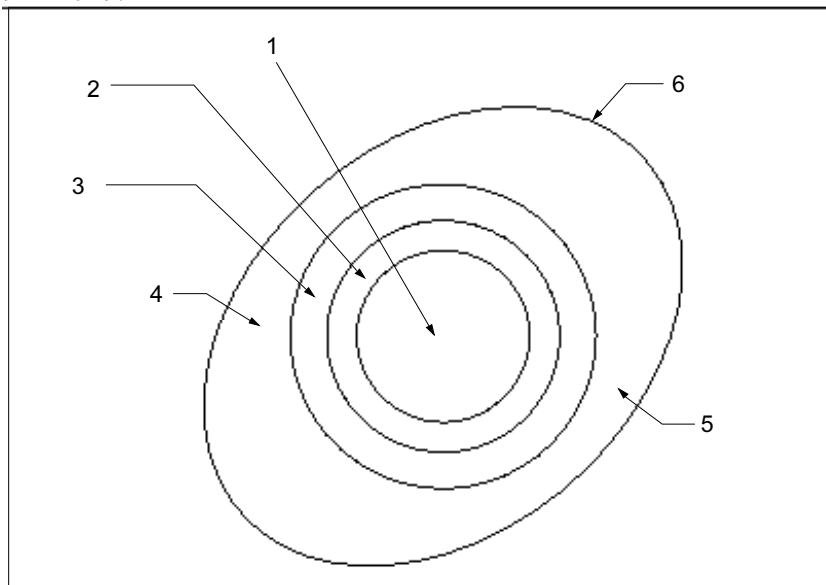
Chaining geometry using centers.

1   Start. Select Circle and pick center.

2   Pick 2.

3   Pick 3.

4   Pick 4. Then select Ellipse and continue.

5   Pick 5.

6   Pick 6.

To Create a Line

Click **Sketch > Line** to open the **Line** menu. Using commands in the **Line** menu, you can sketch the following types of lines continuously as a chain or as individual entities:

- Horizontally or vertically between any two points
- At an angle measured from the x-axis (0)
- Tangent to a curve at its second endpoint
- Parallel or perpendicular to a specified line
- Normal to a curve
- Tangent to an arc or a spline
- Tangent to two circles or splines
- Tangent to a curve and parallel or normal to a reference line

To Create a Circle

Click **Sketch > Circle** to open the **Circle** menu. Using commands in the **Circle** menu, you can create a circle by doing the following:

- Locating its center and defining a point
- Making it tangent to other entities
- Locating its center and defining a point as the tangency with another entity
- Specifying its center and the circle radius or diameter
- Making it tangent to three other entities
- Making it tangent to two other entities, with a specified radius
- Using any three points that lie on the circle

To Create an Arc

Click **Sketch > Arc** to open the **Arc** menu. Using commands in the ARC menu, you can create an arc by doing the following:

- Selecting an endpoint that is tangent to another open entity (that is, not a circle or an ellipse)
- Selecting any three points that lie on the arc
- Using the arc center and two endpoints
- Using the arc center, a start point, and an angle that strokes the arc from the start point in a counterclockwise direction
- Making it tangent to three other entities
- Making it tangent to two other entities, with a specified radius

To Create a Spline

Using the **Spline** command in the **Sketch** menu, you can create a spline by defining the start point and any number of interpolation points.

1. Click **Sketch** > **Spline**.
2. Using the GET SELECT menu, select the start point and then any number of interpolation points to define the

spline.
3. To end the spline at the last selected point, press the middle mouse button.

To Create an Ellipse

Using commands in the Sketch > ELLIPSE menu, you can create ellipses by doing the following:

- Selecting the center, a point to locate the major axis, and an angle that rotates the geometry about the major axis.
- Defining the length and orientation of the major axis and specifying the rotation angle about the major axis.
- Locating the center, a point locating the major axis, and a point locating the minor axis.
- Selecting two points to define the major axis, and a third point anywhere else on the ellipse.
- Using a rotation angle about the major axis. The angle represents what you see of a circle after you have rotated it; the more you rotate it, the less you see of it. For example, if you type 0 degrees, the system creates a circle; if you type 90 degrees, it creates a line.

Example: Creating an Ellipse
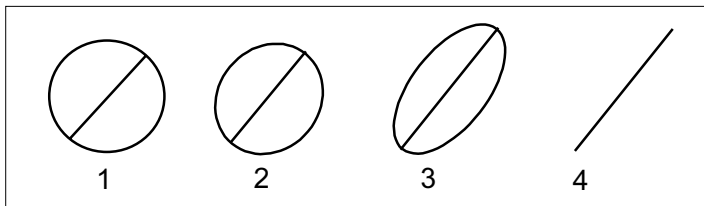
1   0 degrees.
2   30 degrees.
3   60 degrees.
4   90 degrees.

To Create a Fillet

A fillet rounds the corner of a selected line intersection to a given radius.

1. Click **Sketch > Fillet**.

2. Click a point on each line where you want the fillet to start.

3. Enter a radius or accept he default at the prompt line.

To Create a Chamfer

Chamfers are lines that intersect and trim other lines. The system calculates the intersection of the lines, so the lines do not have to actually touch; it applies chamfer dimensions at this intersection point. When it creates a chamfer, it discards the chamfered portion of the lines. Using commands in the Sketch > Chamfer menu, you can create chamfers by doing the following:

- Creating a line between two other lines 45 degrees from the first selected line and of dimension "d" along both selected lines
- Using the same dimension value along both of the selected lines
- Using a different chamfer dimension for each selected line
- Using a chamfer dimension at an angle to the reference entity

Selecting Draft Entities for Sketching

When selecting points for sketching in Pro/DETAIL, you can select draft entities or model geometry. To define the endpoints and centers, use commands in **Abs Coord** to locate a point relative to the current drawing coordinate system.

The GET POINT Menu

- **Pick Pnt** locates a point anywhere on the screen.
- **Vertex** locates a point at the endpoint as well as at the midpoint of an open curve (line, arc, spline), the center of a draft arc or circle, or the intersection between two entities.
- **On Entity** locates a point anywhere along an entity (draft or model). The system places the point on the selected entity closest to where you selected. If you select the end of the entity, it locates the point at the endpoint.
- **Rel Coords** locates a point relative to the last point that you selected using specified x- and y-coordinate values in drawing units.

Tip: Selecting Points for Sketching

Use the configuration file option `draw_points_in_model_units` to define the current draft view's coordinate values as model units rather than drawing units. The GET POINT menu uses the scale of the draft view and the draft view's model units for relative and absolute coordinate entry and display in the message area.

To Select Several Draft Entities

You can select several draft entities for modification simultaneously by choosing **Pick Many** from the GET SELECT menu, followed by **Pick Box**. If you choose **Inside Box** from the submenu, you can only select items that are completely within the box. If you choose **Across Box**, you can select all draft entities that are partly *or* completely within the box. Use the left mouse button to specify opposite corners of the box, press the middle button, and press the left button to complete the action.

Tip: Selecting Several Draft Entities

If you set the configuration file option `dwg_select_across_pick_box` to `yes`, the system highlights the **Across Box** command by default when the PICK MANY menu appears. If you set this option to `no`, it highlights the **Inside Box** command by default.

About Draft Dimensions

Draft dimensions and the geometry that they reference are completely independent of each other, unless associative dimensioning is in effect. You can make the dimensions of draft entities associative—so that the system updates them to reflect changes in draft entities—by setting the drawing setup file option `associative_dimensioning` to `yes` (the default value). The system considers the parent/child relationship between draft entities and their associative dimensions, and performs the following procedures on them:

- Deletion
- Switching to another drawing sheet
- Translation
- Rotation
- Rescaling, including change of drawing format size
  Notes:
  The system assigns associativity status to new dimensions only.

Draft dimensions are driven dimensions. When the scale of draft entities changes, the values of their associative dimensions also change.

If you select a draft entity for any of the preceding procedures, you implicitly select all of its dimensions. For all procedures except deletion, if you select a dimension, you also select the parent entity and all of its other dimensions. If you have implicitly selected a dimension by selecting an entity:

- Unselecting the entity unselects the dimension as well, unless the dimension was also selected as another entity's dimension, or by itself.
- You cannot unselect the dimension until you unselect the entity itself.

Pro/ENGINEER considers draft groups for the preceding procedures to be parents of *all of the dimensions of their entities*.

Draft Scale Independent of View Scale

Draft scale is completely *independent* of view scale. Although a draft entity associated with a view appears as a different size on the screen, when the scale of that view changes, the dimension values of the draft entity do not change in any way.

Examples: Drawing Format Change and Moving Draft Entities

The following figure shows that when you reduce the size of the drawing by half, the system scales the views and draft entities to fit, although they are the same size on the screen. The text height does not change, but the system halves the draft cross section's dimension value.

Effects of Drawing Format Change on Associative Draft Dimensions
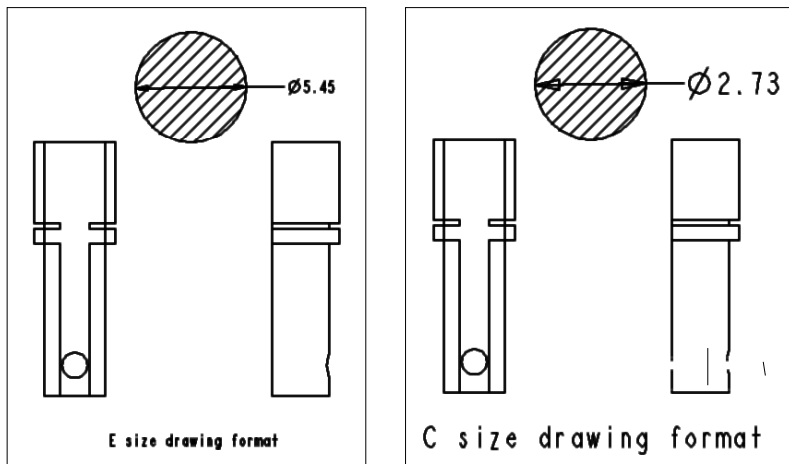
Moving Draft Entities Without Associative Dimensioning
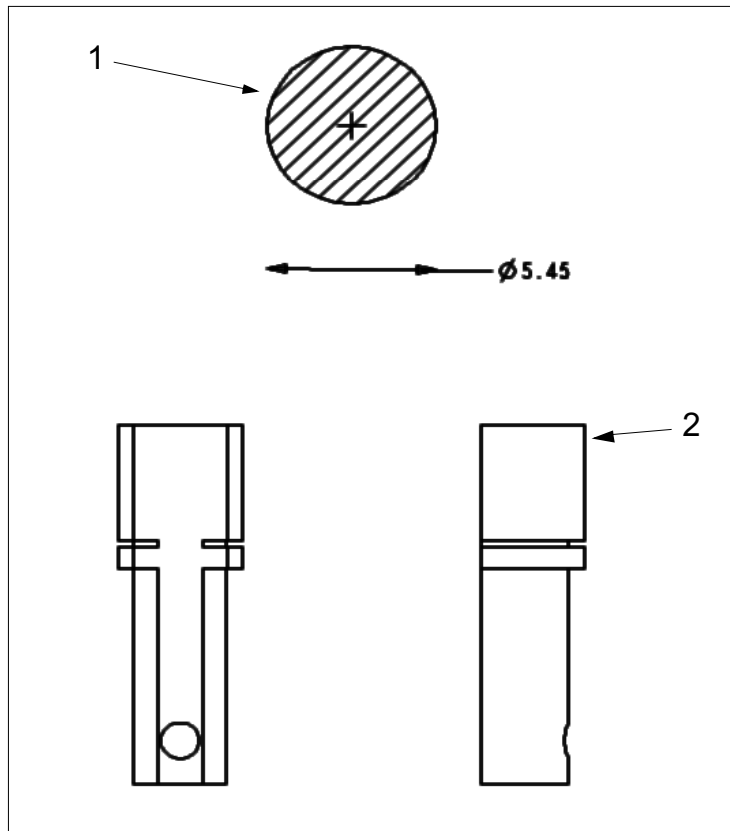
1   The system translates the draft cross section vertically, but since associative dimensioning was not set, it
    leaves behind the diameter dimension.
2   Model view.

About Obtaining Geometry Information

To identify draft geometry on a drawing, choose **Drawing** from the Pro/ENGINEER **Info** menu, followed by
Highlight by Attributes. To obtain information about specific draft entities, choose **Measure Draft Entities...**
from the Pro/ENGINEER **Info** menu.

To Measure a Distance

To measure the distance between two parallel lines, two points, or a point and a line, choose **Distance** from the
Type container; then use the selection tools in the **Definition** box to select two entities. In the **Results** box, the
system displays the distance measurement.

To Measure an Angle

To measure the angle between two selected lines, choose **Angle** from the **Type** container; then use the selection
tools in the **Definition** box to select two lines. In the **Results** box, the system displays the angle measurement.

To Determine the Slope of Two Entities

To determine where the slope of two selected entities are equal, choose **Tangent Point** from the **Type**
container; then use the selection tools in the **Definition** box to select the entities. In the **Results** box, the system

displays the angle of slope at the tangency points, the distance between the two tangency points, and the coordinates of the tangency points.

To Determine the Intersection Point of Two Entities

To determine the intersection points of two selected entities, choose **Intersection Point** from the **Type** container; then use the selection tools in the **Definition** box to select the entities. In the **Results** box, the system displays the angle of slope at the

intersection point for both entities and the intersection coordinate values (to the current drawing coordinate system).

About Showing Dimensions in a Drawing

Using the View > Show and Erase dialog box, you can display model dimensions in a drawing as well as show dimensions for a range of features. To show dimensions in an isometric view, set the drawing setup file option `allow_3D_dimensions` to `yes`. The location of displayed model dimensions depends on the view orientation. Note the following special cases:

- When the sketching plane of an extruded or revolved section is neither parallel nor normal to the screen, the system still shows the linear dimensions of the section that are parallel to the screen.
- For clipped views, Pro/ENGINEER rotates the dimensions of a revolved section up to 180 degrees to bring them into the view outline.
- When the system cannot show the dimensions of a normal-to-trajectory sweep on the sketching plane, it shows them on the other end or, if that is impossible, on some intermediate plane.
- Pro/ENGINEER does not show the dimensions (in a view) of features that you have suppressed using **By View**. If possible, it displays them in another view.

To Display Model Dimensions in a Drawing

1. Click **View >Show and Erase**. The **Show/Erase** dialog box opens.
2. In the **Show/Erase** dialog box, click **Show**.
3. Click **Dimension** or **Reference Dimension** in the **Type** box.
4. Select a button in the **Show By** box to specify where to show the dimensions in the drawing.
5. Click **Preview** to view the changes. Select **With Preview** to specify the items to show.
6. Click **Close**.

To Specify Dimensions for a Range of Features

1. Click **View >Show/Erase**. The **Show/Erase** dialog box opens.
2. In the Show/Erase dialog box, click **Show**.
3. Click **Dimension** in the **Type** box and **Feature** (or **Feat & View**).
4. Choose GET SELECT > **Sel By Menu**. Select the model for which you want to show the range of feature dimensions.
5. Choose SPECIFY BY > **Number Range**.
6. Type the lower limit and upper limit of the range of feature numbers.

Displaying Dimensions in Detailed and Partial Views

A dimension that references nonsolid geometric features (axes, datum points, datum planes, and so forth) can be present in a detailed or partial view only when the following requirements are satisfied:

- At least one of the entities being dimensioned must be within the spline boundary.
- This entity must also be within the view boundary of the view—the view boundary is defined by the solid geometry of the part. (Nonsolid geometric features are not considered to be solid geometry.)

Check that these requirements are met before you change a view to a detailed or partial view, or dimensions will

disappear from the new view.

The drawing setup file option `clip_dimensions` affects the display of dimensions in detailed views. When you set it to `yes`, dimensions that are completely outside of the view boundary do not appear. Dimensions that cross the view boundary appear, and the leader that seems to have no geometric reference appears with a double arrow (>>). To change the arrow style, set the default arrow style of clipped dimensions by specifying a value for the drawing setup file option `clip_dim_arrow_style` (`double_arrow` is the default).

Showing Dimensions in Assembly Drawings

In assembly drawings, you can display the parameters of assembly features and all assembly components; however, you cannot show the dimensions of subassemblies. The dimensions in an assembly drawing are visible only at the assembly level. You must have the assembly from which a drawing was created in session in order for the dimensions to appear for modification.

Note: When you use automatic replacement to replace an assembly model with another model of the same family, the equivalent dimensions appear if the system shows dimensions in the assembly drawing on a component that you replace with the new instance. If you use manual replacement, it does not preserve the dimensions.

To Show Dimensions in an Assembly Drawing

1. Click **View > Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show** and then click **Dimension** in the **Type** box.
3. To display the dimensions of a part without showing the dimensions of a merged feature, do one of the following:
   - Select **Part** from the Show/Erase dialog box and select the part on the screen.
   - Choose GET SELECT > **Sel By Menu** to select dimensions of the original part from the namelist menu.
4. To display the dimensions of a merged feature in an assembly drawing, select **Feature** in the dialog box and choose GET SELECT > **Query Sel** to select the feature of the part that was used for reference or copy.

To Erase Dimensions from a Drawing

You can remove dimensions from the display by erasing them, but this does not delete them from the model. Pro/ENGINEER does not recall the location of a dimension after it erases it. If you show the dimension again, it might not appear at the same location as before.

1. Click **View > Show and Erase**. The **Show/Erase** dialog box opens.
2. In the **Show/Erase** dialog box, click **Erase**.
3. Click **Dimension** or **Reference Dimension** in the **Type** box.
4. In the **Erase By** box, select one of the following:
   - **Selected Items**—Erases specified dimensions.
   - **Feature**—Erases dimensions in the selected feature.
   - **Feat & View**—Erases dimensions in the selected feature that belong to the selected view.
   - **Part**—Erases dimensions associated with a part at the selected assembly location.
   - **Part & View**—Erases dimensions associated with a part in a view.
   - **View**—Erases all dimensions in the selected view.
   - **Erase All**—Erases all dimensions in the drawing.

To Show and Erase View Notes

To erase and show view notes (detail view name, view scale, cross section name, graph feature view name), use the Show/Erase dialog box.

To Show Model Notes in a Drawing

Using the Show/Erase dialog box and (optionally) the Model Tree Window, you can show model notes in drawings. Their location within the drawing is dependent upon their actual placement in the model. Unplaced notes appear in a default location when you show model notes in the drawing. You can use the text of a model note to create a drawing note by reading in a model note from a file.

Model Notes in Drawings

When working with model notes in drawings, consider the following:

- Once you show a model note in a drawing, the system does not make changes to its attachment or location if you change either in the model. Also, if you make any changes to the note's attachment or location in the drawing, the system does not make those changes in the model.
- You cannot access drawing-level notes in models.
- When you modify the contents of a model note in a drawing, the model changes.
- To disallow modifications to model notes, you must set the configuration file option `draw_models_read_only` to `yes`.

About Creating Dimensions

In Drawing mode, you can create the following types of dimensions:

Driven Dimensions

Driven dimensions are derived directly from the model dimensions as laid out in sketcher. Changes in the model *drive* their values as shown in the drawing.

When working with driven dimensions, keep in mind the following:

- Driven dimensions appear in the drawing in which they were created, but the system stores them with the model. You cannot see them in other drawings, or in Part or Assembly mode.
- You can modify the number of digits (decimal places) in driven dimension values, but the rounded dimension value does *not* drive the model geometry. Driven dimensions do reflect changes to part or assembly geometry, but you cannot modify their values from the drawing.

Draft Dimensions

A dimension shown between two draft entities (for example a sketched object or construction line) or between a model object and a draft object is called a Draft dimension. The value is based on the *draft scale* (specified by using the drawing setup file option `draft_scale`) and the actual drawing sheet units and size.

Draft scale is the relation between what the entity size appears to be versus its actual dimension. For example, assume that an edge of a part is 4 inches long:

- If the draft scale is 1.0, the entity appears to be 4 inches in the drawing.
- If the draft scale is 4.0, the entity appears to be 16 inches in the drawing (it appears to be four times larger than it actually is).
- If the draft scale is 0.5, the entity appears to be 2 inches in the drawing (it appears to be one-half the size it actually is).

Note: Regular nonassociative draft dimensions do not change with draft scale.

Associating a dimension with a draft entity links the two together so that the dimension reflects changes to the entity. When you have made a draft dimension associative (using the drawing setup file option `associative_dimensioning`), the draft dimension's value changes if you change the draft scale, move the entity, and trim and intersect the entity. The system reflects the changes to the draft scale in the value of the draft dimension when it regenerates the drawing and creates new draft entities at the scale value.

Reference Dimensions

While in Drawing mode, you can create reference dimensions using model geometry in a drawing view. The system saves dimensions with the model and associates them with the view in which they were created. All reference dimensions have the notation "REF" to distinguish them from other dimensions. The system encloses them in parentheses if you set the configuration file option `parenthesize_ref_dim` to `yes`.

Saving Dimensions to the Part or to the Drawing

When you create dimensions in Drawing mode, the configuration file option `create_drawing_dims_only` determines whether the system saves them in the associated part or in the drawing itself.

When set to `yes`, it saves all new dimensions created in the drawing in the drawing only.

When set to no, (the default), it saves all new model dimensions (not draft dimensions) created in the drawing to the associated part or assembly. Draft dimensions are still saved to the drawing.

The setting should depend on your particular work scenario. If you are using Intralink, if the dims are stored in the model, the model will be marked as modified and will have to be re-submitted back to intralink. To avoid this every time you reference a model for drawing, you can set the option to yes.

Alternately, if you want to use a set draft datum in a gtol attached to a dimension, the dimension has to be stored in the model, so you would set the option to no.

To Create a Driven or Reference Dimension

1. Click **Insert > Dimension** or **Reference Dimension**. (*Dimension* is the driven dimension.) Use the fly-out menus to specify use of **New References**, a **Common Reference**, or an **Ordinate** dimension. The **ATTACH TYPE** menu appears on the Menu Manager.
2. Select an edge, an edge and point, two points, or a vertex using the ATTACH TYPE menu commands:
   - **On Entity**—Attaches the dimension to the entity at the pick point, according to the rules of creating regular dimensions.
   - **Midpoint**—Attaches the dimension to the midpoint of the selected entity.
   - **Center**—Attaches the dimension to the center of a circular edge. Circular edges include circular geometry (holes, rounds, curves, surfaces, and so forth) and circular draft entities. If you select a noncircular entity, attaches the dimension to the entity, as does **On Entity**.
   - **Intersect**—Attaches the dimension to the closest intersection point of two selected entities.
   - **Make Line**—References the current x- and y-axes in the orientation of the view of the model.

   Note: To create an arc-length driven dimension, select the ends and the middle of an arc; then choose ARC DIM TYPE > **Arc Length**.
3. Locate the dimension. If you have selected two arcs or circles (the system highlights each arc or circle in turn), use the ARC PNT TYPE menu to do one of the following:
   - Choose **Center** to create a dimension between the centers of the arcs, ellipses, or circles.
   - Choose **Tangent** to create a dimension between the circle, arc, or ellipse edges—the tangency closest to where you selected.
4. From the DIM ORIENT menu, choose one of these commands:
   - **Horizontal**—Reflects the horizontal distance.
   - **Vertical**—Reflects the vertical distance.
   - **Slanted**—Reflects the shortest distance between two attachment points (available only when the dimension is attached to points).
   - **Parallel**—Reflects the distance in the direction parallel to another entity.
   - **Normal**—Reflects the distance in the direction normal to another entity.

Using a Common Reference

Use **Common Ref** to create standard dimensions from a common reference. However, to use a common reference, the last dimension that you created must have more than one reference object. If, for example, the last dimension that you created is an angular dimension measuring the spanning angle of an arc, there is only one reference object (the arc), so the system does not use it as the first reference for the next dimension. The same condition applies to draft entities.

To Create a Standard Dimension from a Common Reference

1. Click **Insert > Dimension** or **Ref Dim** > **Common Ref.**
2. The system highlights the first reference of the last dimension that you created and uses it as the first reference of the next dimension that you create.

Example: Creating a Standard Dimension from a Common Reference
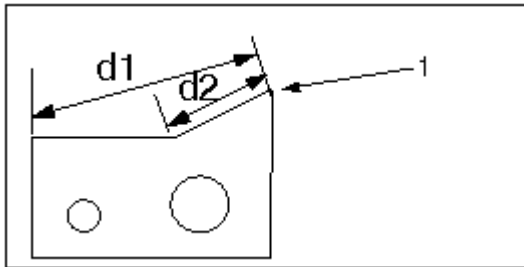
1   A common reference.

Using Driven and Reference Dimensions in Relations

You can use driven (or "ad" type) and reference dimensions on the right side of relations only (to drive the value of other dimensions or parameters). If you have used a driven dimension in a relation to drive a model dimension, the model dimension also changes until you regenerate the model in Drawing mode. You can use reference dimensions to drive model dimensions because they regenerate in Part or Assembly mode; however, you should not use driven dimensions to drive model dimensions because incorrect model dimensions might occur when you change the model in Part or Assembly mode. To add relations, use the **Relations** command in the DWG SET UP menu. Driven dimensions can reference a coordinate system.

Adding Jogs to Ordinate Dimensions

You can add a jog to an ordinate dimension when you are creating a baseline reference; however, you must choose this command before selecting the dimension for modification. You can also add a jog to an existing ordinate dimension.

To Create a Jog

1. Click **Edit > Lin to Ord**.
2. Select the dimension to convert from linear to ordinate. The dimension disappears from the screen.
3. Click **One Jog**.
4. Make two pick points corresponding to the first and second knuckle in the ordinate dimension witness line.

The system always locates the dimension text the same distance from the model as it was in the linear dimension, so you should select points along the witness line and between the dimension text and the model.

To Create a Coordinate Dimension

1. Click **Insert > Coordinate Dimension…**.
2. Use the **GET SELECT** menu to select model geometry to which to attach the coordinate dimension leader.
3. Select an edge, entity, or axis where the coordinate dimension is attached.
4. Select the location of the dimension symbol.
5. Select the appropriate x- and y-dimension. The system creates a coordinate dimension with these values.
   Note: You cannot select ad, dd, and ordinate dimensions as coordinate dimensions.

To Move a Dimension Between Views

1. Select a dimension to move. To select several dimensions at one time you can use **Pick Many** in the GET SELECT menu.
2. Choose **Edit > Switch to View**.
3. Select a view.
Notes: If Pro/ENGINEER cannot display a dimension in the selected view, it issues a warning and does not move it.

When you select a patterned feature dimension, all dimensions of the pattern move.
When you switch the view of an ordinate baseline, all ordinate dimensions that reference it also switch views.

To Modify the Arrow Style of Leaders

Use **Arrow Style** to modify the arrow style of leaders for dimensions, notes, 3-D notes, geometric tolerances, symbols, or balloons. For example, you can change an arrow to a dot or a box.

1. Click **Format > Arrow Style**.
2. Select the arrow of the entity to modify, and choose one of the arrow styles from the ARROW STYLE menu.
   Note: The drawing setup file option draw_arrow_style controls arrow style for all detail items involving arrows, including leaders of dimensions, notes, 3-D notes, geometric tolerances, symbols, and balloons.
   The drawing setup file option dim_dot_box_style controls the display (filled or hollow) of dots and boxes only for leaders of linear dimensions.
You can also manipulate dimension leader arrows as follows:

3. Set the default arrow style of clipped dimensions by specifying a value for the drawing setup file option clip_dim_arrow_style (double_arrow is the default).
4. Switch their direction by choosing **Flip Arrows** from the right mouse button shortcut menu.
   Note: You cannot modify the arrow style of angular, diameter, and radial dimensions.

To Create Dimensions Without an Elbow

1. Set the drawing setup file option default_dim_elbows to no.
2. Create the dimension in the drawing. The system assigns the dimension an elbow with a default value of zero and does the following:
   - Centers the line of text containing the dimension value on the end of the leader line.
   - Trims back the leader line to the box surrounding all of the lines of dimension text.
   - Pads the character height by half.
   - Displays the text center-justified about the endpoint of the dimension line.

   Note: A dimension with a leader that does not have an elbow appears different, with its text center-justified (this is the default justification).

To Flip Dimension Extension Lines

1. Choose **Move**; then select dimension text.
2. Move the dimension text between the extension lines.
3. Move the dimension to the opposite side.

To Modify a Dimension So the Dimension Symbol Always Appears

1. Select the dimension.
2. Click **Edit** > **Properties**. The **Modify Dimension** dialog box opens.
3. Click the Dim Text tab. Replace the symbol "@D" that shows the dimension value with "@S" to show the dimension symbol. Any text that was previously added to the dimension is lost.

Note: To change the dimension back to normal display, use the preceding procedure and modify @S back to @D or @O (for drawing dimensions that show text instead of a dimension value).

You can replace the symbol without invalidating relations where the dimension appears.

About Dimension Types

Dimensions can be standard or ordinate.

- Standard dimensions are used to describe a dimension between two selected points. They use two witness lines and a leader.

- Ordinate dimensions are used to describe the dimensions between several points and a common base reference. They use a single witness line with no leader. All dimensions that reference the same baseline must share a common plane or edge. To create an ordinate series you must first convert a standard dimension, and select one of its witness lines as the baseline.
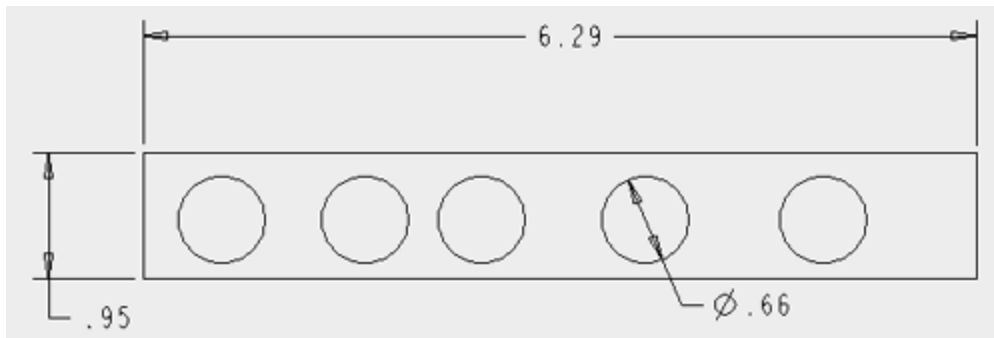
Standard Dimensions
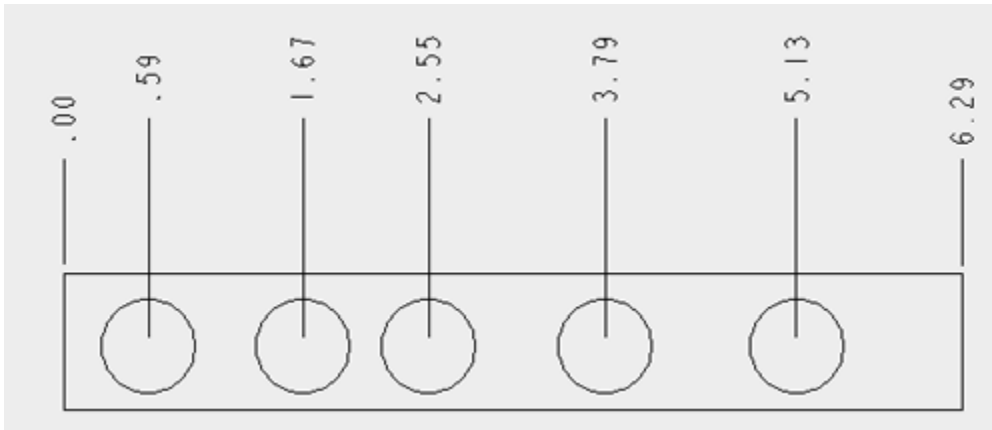
Ordinate Dimensions

About Working with Ordinate Dimensions

You can modify linear dimensions so that they appear as ordinate dimensions. When you change dimensions to an ordinate type in a drawing, they appear the same way in Part or Assembly mode, and in any other drawing in which the model appears. Ordinate dimensions use a single witness line with no leader, and are associated with a baseline reference, shown in the next figure as ".00". All dimensions that are to reference the same baseline must share a common plane or edge. You can then use the common plane or edge for a baseline reference.

Example: Drawing with Ordinate Dimensions
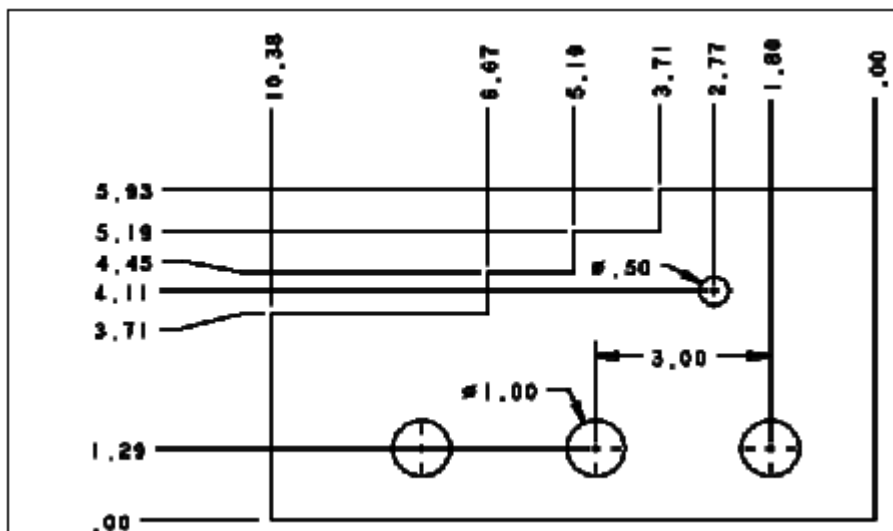
Changing from Linear to Ordinate

To modify a dimension type from linear to ordinate, you must first establish a reference baseline. For any set of related dimensions (dimensions that share a common plane or edge), you need to create only one baseline reference. If you just created a baseline, it remains set until you set another, or until you exit the MOD DIM TYPE menu. Only one baseline reference can be current (set) at one time. To set another, choose **Set Base** and select the existing baseline reference that you want to set. You can select dimensions that have witness lines coincident with the set baseline and convert them from linear to ordinate dimension type.

When converting the dimension, you can also add a jog to the witness line to improve the spacing of the dimensions.

To Convert a Linear Dimension to an Ordinate Dimension

The dimension to be converted to ordinate must be shown as linear. The following dimensions cannot be converted to ordinate:

- – A diameter dimension shown as linear
- – A centerline dimension

1. Click **Edit > Linear to Ordinate**.
2. Select the dimension to convert.
3. Select the witness line to be the baseline reference. The dimension changes automatically to an ordinate type dimension, indicated with a value of .00.
4. Select linear dimensions. The system places each one at the end of the witness line that is not coincident with the current baseline.

Notes:

- You cannot select dimensions that do not share the *same* baseline reference.

- When you switch dimensions from linear to ordinate, the baseline reference regenerates. However, if an orphan baseline (one that does not have dimensions associated with it) has driven children (dimensions), the baseline and the children do not regenerate. If many driven dimensions in a drawing reference an orphan baseline, modify the dimensioning scheme by selecting a side from which to attach the baseline. The baseline then regenerates at the correct location. To update children, choose **Regenerate** from the DRAWING menu, followed by **Draft**.

- If you attempt to create associative dimensions using a nonassociative baseline, the system asks you to confirm that you want to create associative ordinate dimensions using the baseline reference.

- Baseline references appear on the drawing or in the model as ".00". Using the Show/Erase dialog box, you can erase them from the drawing for cosmetic reasons. Also, you can select and delete baseline references that are no longer necessary (if ordinate dimensions are not associated with them).

Showing Linear Dimensions as Ordinate

You can show dimensions or reference dimensions as linear or ordinate dimensions.

- If the system cannot show a dimension as ordinate from the selected baselines, it does not show it at all.

- If it shows dimensions as ordinate on the drawing, it shows them as ordinate on the model as well.

Typically, Pro/ENGINEER does not show negative or positive values for dimensions unless you set the configuration file option `show_dim_sign` to `yes.` However, if you create dimensions which reference features offset from a coordinate system, or assign a negative value to them, the negative value appears even if you have set this option to "no."

To Change Ordinate Dimensions to Linear

When you convert an ordinate dimension to linear, the dimension loses its association with the baseline reference. If you convert the dimension back to an ordinate dimension, you can select any appropriate baseline reference.

1. Click **Edit > Ordinate to Linear**.
2. Select any ordinate dimension to convert.

To Convert Diameter Dimensions to Linear

You can convert a diameter dimension to a linear dimension, or convert a linear dimension for a diameter to a

diameter dimension without switching the view. When you switch views on one of these converted dimensions, Pro/ENGINEER tries to display the dimension in the format type prior to the conversion, then in the converted format type, and finally in its original displayed view.

1. Select the diameter dimension to convert to linear.
2. Click Edit > Properties. The Modify Dimension dialog box opens.
3. Select the **Show as linear dimension** check box and then click **OK**.
Note: You cannot convert a linear dimension (for example, the depth of an extruded slot) to a diameter dimension.

Creating and Deleting Ordinate Driven Dimensions

When creating ordinate driven dimensions, you must create a baseline in the required direction by converting a standard dimension into an ordinate dimension, and then create ordinate dimensions using the baseline reference.

When creating ordinate driven dimensions, keep in mind the following:

*   You cannot create ordinate dimensions from a baseline dimension when the configuration file option create_drawing_dims_only is set to yes and a dimension with a d or ad type symbol (denoting either a driving model dimension or an associative draft dimension) is the active ordinate baseline.
*   To control the orientation of the ordinate dimension text, set the drawing setup file option orddim_text_orientation to either parallel or horizontal.
*   If you set the drawing setup file option ord_dim_standard, you can connect ordinate dimensions measured from one baseline with a line. When extension lines are interconnected, all related dimensions move when you move one.
*   You cannot switch ordinate driven dimensions to another view.

To Create Ordinate Dimensions

Before you can create ordinate dimensions you must have a standard dimension defined on the baseline object. You convert one of the dimension's witness lines into the baseline for the succession of ordinate dimensions.

1. Click **Insert > Dimension > Ordinate.** In the **Menu Manager** click **ORD MAKE** > **Create Bases**.
2. Select the standard dimension to convert into an ordinate dimension. The dimension and witness lines are highlighted.
3. Select the witness line to be the baseline. The ordinate dimension is created, and the other witness line is dimensioned from the base.
4. Click ORD MAKE > Create Dims. Select the next edges to dimension. The baseline remains selected until after you place the dimensions. You can create more than one ordinate dimension at a time.
5. Once you have finished selecting edges, click the middle mouse button. The system places the dimensions.

To Delete Driven Ordinate Dimensions

You can delete driven ordinate dimensions, or those created by using **Ordinate to Linear**.

The process for deleting a baseline for ordinate driven dimensions is different from the process for deleting the baseline when you have converted dimensions with a common witness line from linear to ordinate. You cannot delete the baseline and the dimension that was converted to create the baseline until you convert them to a linear dimension.

To Delete the Baseline

1. Click **Edit > Ordinate to Linear**.
2. Select the ordinate dimension that corresponds to the baseline, but not the baseline itself. The dimension converts from ordinate to linear.
3. Return to the menu bar and click **File > Delete**.

4. Select the linear dimension (that was previously assigned to be the ordinate dimension baseline) for deletion.
5. Choose **Done Sel**. A ".00" reference appears for any remaining ordinate dimensions.

To Reroute Dimensions with Lost References

References to dimensions are lost when modifications or additions are made to features. Lost references occur when parts are suppressed, during feature redefinitions or creations, and so forth. Features in a model change and driven dimensions on the drawing no longer reference the same feature. You can identify lost references in a drawing by the color magenta.

1. Click the lost dimension to reroute and then use the right-mouse button to see a shortcut menu.
2. Click **Reroute Attach** to reroute dimensions to new references.

To Automatically Dimension Radial Patterns

To automatically dimension a radial pattern feature (such as a bolt circle):
3. Click **Insert** > **Dimension** or **Ref Dim** > **Common Ref**. The **Attach Type** menu opens on the Menu Manager.
4. In the **Attach Type** menu click **Center**.
5. Select the axis of one of the pattern features. You are prompted to select additional entities to dimension.
6. Select the base point to dimension against. (For example, you can select a csys at the center of the pattern or an entity outside of the pattern.)
7. Click **Done Sel**. The **Dim Orient** menu appears.
8. Select an orientation. You are prompted: "Create Dims to the other members of the feature pattern?"
9. Click Yes. The remaining pattern members are dimensioned.

About Symbols

Symbols are collections of draft geometry and text. When you use them in a drawing, they become single entities or instances. You can add as many instances of a symbol as you like.

Accessing Symbols

Pro/ENGINEER symbols exist in two different areas: the user-defined symbols area and the system symbols area. The *user-defined symbols area* is the default storage area for user symbols. The *system symbols area* is read-only and contains system symbols provided by Pro/ENGINEER with the Pro/DETAIL module (such as the Welding Symbols Library).

To Set the User-Defined Symbols Area

To specify the area in which you want to store user-defined symbols, set the configuration file option `pro_symbol_dir`. This option automatically creates a search path to the specified directory. Pro/ENGINEER saves all symbols into and retrieves them from this directory by default if you add this option to your configuration file. If you change the value, the system does not delete symbols used in the drawing; once you add them, it stores the definitions locally in the drawing.

You should establish a single directory as your *user* library for all standard symbols. If you do not specify one, the system searches in the current working directory. You can change the default area for user-defined symbols by entering a new value for the `pro_symbol_dir` configuration file option. When you change this directory, you do not have to modify the configuration file; however, this change is valid only for the current Pro/ENGINEER session. Use this option to define new symbols that you store in a local or temporary directory; you can still easily retrieve symbols from the standard symbols area.

To Use the System Symbols Area

The system symbols area contains libraries of Pro/ENGINEER symbols that are available with the Pro/DETAIL module. This area is *read-only*. To retrieve a symbol in Drawing mode, click **Insert > Symbol Instance**. In the **Symbol Instance** dialog box, click **Retrieve**. The Open dialog box then opens, and you can retrieve a system symbol from disk.

The Welding Symbols Library, for example, provides a collection of generic system symbols according to the ANSI standard, and a collection of symbols according to the ISO standard. Using this library, you can create a variety of welding, brazing, and examination symbols in a drawing. Before you create an instance, familiarize yourself with the procedure for adding instances.

To Create Your Own Library

You can create your own symbols library. For information on how to create and locate your own library of symbols, see the *Pro/ENGINEER Installation and Administration Guide*.

To Change the Symbol Directory

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **Symbol Dir**.
3. Type the pathname of the directory for the new `pro_symbol_dir`.
4. To restore the default pathname in the current session, do one of the following:
   – Repeat this process and substitute the original pathname.
   – Choose **Utilities > Options** from the Pro/ENGINEER menu bar and reload the configuration file in which this option resides.

Simple and Generic Symbols

Pro/ENGINEER supports two kinds of symbols—simple and generic. A *simple* symbol is a symbol instance that is identical to the symbol. A *generic* symbol defines a family of similar symbols; it contains all entities pertaining to this family. You can arrange geometry and text in a generic symbol in groups and subgroups, creating a tree structure of symbol definition.

To Delete a Symbol Definition

Using the **Delete** command in the DWG SYMBOL menu, you can remove a symbol definition and all of its instances from a drawing.

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **Delete**.
3. Specify the symbol by choosing **Name** or **Pick Inst** from the **GET SYMBOL** menu.
4. Type [Y] to remove the symbol definition and its instances.

Storing Symbols

You can store symbols in a specified area by setting the configuration file option `pro_symbol_dir`. If you do not specify a path, Pro/ENGINEER stores symbols in the working directory. When storing a symbol, you can enter an offset path that branches off from `pro_symbol_dir`. For example, for a UNIX-based system, if you have specified "pro_symbol_dir" as */usr/proe/symbols*:

- If you press ENTER, the system places the symbol in */usr/proe/symbols*.
- If you type [`down_one_dir`], it stores the symbol in */usr/proe/symbols/down_one_dir*.
- You cannot go up the directory tree by typing [`..`].
- If you want to store the symbol in a directory that you cannot access as an offset of the current

pro_symbol_dir, change pro_symbol_dir before you begin.

- You do not have to store the symbol in order to continue using it in the drawing. However, if you do not write it to disk, the system only stores it locally in the drawing and does not make it available for use in other drawings or by other users.

To Store a Symbol

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **Write**.
3. Specify the symbol by choosing **Name** or **Pick Inst** from the GET SYMBOL menu.
4. Type the offset directory path from the directory specified by pro_symbol_dir in which to store the symbol.

Redefining Symbols

Using the **Redefine** command in the **DWG SYMBOL** menu, you can add, move, delete, or change detail items composing a symbol.

You can modify a symbol by doing the following:

- Create new entities composing a generic symbol.
- Modify existing entities composing a generic symbol.
- Change the attachment style.
- Change group definition by adding or removing entities from groups, or deleting one or all groups.

Effect of Symbol Redefinition on Instances

When you redefine a symbol, it affects the display of all subsequent instances and all symbol instances that you have added to a drawing using the **By Reference** command.

Updating Symbols Used in Drawings

When you place a symbol that you have redefined, it is possible, but not necessary, to update the symbol in the drawing to reflect the most recent definition. The system prompts you to update a symbol when the version of a retrieved symbol is later than the version of the symbol with the same name on the drawing.

Updating the drawing symbol changes every symbol in the drawing with that name. If you do not update the symbol in the current drawing, any additional instances of the symbol that you create are of that version, not the most recent one. To update an existing symbol, choose **Retrieve** from the **GET SYMBOL** menu, and select its name in the existing drawing. Type [Y] to update all instances of the symbol in the current drawing to the most recent version.

To Redefine a Generic Symbol

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **Redefine** and then select a symbol instance.
3. The symbol edit window displays the generic symbol. Create, delete, or modify any of the symbol entities by choosing an appropriate editing command from the **SYMBOL EDIT** menu:
   - **Detail**—Creates, deletes, moves, translates, and performs other procedures on items composing a symbol in a way similar to manipulating draft entities in drawings.
   - **Mod Text**—Changes text.
   - **Mod Xhatch**—Modifies cross-hatching immediately upon its creation in the symbol edit window. After you exit the window, it becomes a collection of individual entities.
   - **Import**—Imports items into the Pro/ENGINEER symbol edit window.
   - **Attributes**—Respecifies a leader type, if any, and symbol size characteristics.
     If you have created new entities, they automatically belong to the uppermost level and appear in all

instances. If you have deleted an entity, the system automatically removes it from all groups and subgroups to which it previously belonged.

4. To add the entity (that you have just created) to a group, use the group editing commands.
5. Choose **SYMBOL EDIT > Groups**.
6. Choose **SYM GROUPS > Edit**. The TOP LEVEL menu displays the top-level groups. Choose a group to indicate the branch that you want to follow. The system highlights in blue all entities associated with this group.
7. To add the entity, choose **EDIT GROUP > Add**, and select the entity. Confirm the selection by choosing **Done Sel**.
8. To pass down this entity to a lower level group, choose **Change Level**. Select the top-level group with which you have just associated the entity. Only entities belonging to the selected group appear in the symbol edit window, while the namelist menu lists subgroups of the selected group. This indicates that now you are one level down.
9. Choose **Edit**. Choose the desired subgroup from the namelist. Choose **Add**, and select the entity. This associates the entity with the subgroup you have indicated.
10. If you want to go further down, *repeat Steps 7 and 8*.
11. After you finish the editing process, you can respecify attributes. When you have redefined the symbol, you can store the new symbol definition by choosing **DWG SYMBOL** > **Write**. The system creates a new symbol file with the version number to retain the old symbol definition file.
    Note: When you redefine a symbol, any cross-hatching that exists in the symbol is no longer a group, so you should manipulate it accordingly.

To redefine a simple symbol, follow Steps 1 through 6 of the preceding procedure for generic symbols; then choose **Attributes** from the SYMBOL EDIT menu to respecify attributes. Store the symbol by choosing **Write** from the DWG SYMBOL menu.

About Simple Symbols

To define a simple symbol, you must specify the symbol entities (geometry), attributes, origin or attachment point, and any variable text needed. Using the SYMBOL EDIT menu, you can define a simple symbol using draft entities in a format or file.

To Define a Simple Symbol

1. Create the draft geometry and add notes to include as fixed or variable text in the symbol. The text size and placement should be proportional to the geometry. Symbol text and geometry remain proportional when you modify the symbol height.
2. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
3. Click **Define**.
4. Type a name that an existing symbol is not already using (in the directory where you are going to store the symbol) unless you intend to overwrite the existing symbol. The symbol edit window appears.
5. Choose **Copy Drawing** to copy entities from the format or drawing into the symbol edit window. Click **GET SELECT > Pick Many** > **Pick Box** to select all entities quickly.
6. Choose **GET SELECT > Done Sel** to confirm your selection. All selected entities appear in the symbol edit window.
7. Choose **SYMBOL EDIT > Attributes**.
8. Select the desired attributes from the Symbol Definition Attributes dialog box; then click **OK**.
9. Choose **SYMBOL EDIT > Done**. The system confirms that it has successfully defined the symbol.

Example: Selecting Origins for Symbol Placement
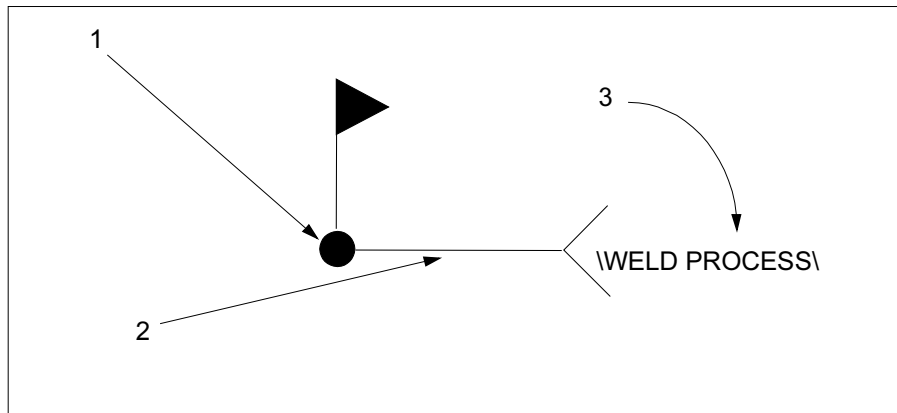
1 Select the center as the origin for the Left Leader.
2 Select anywhere as the origin for No Leader.
3 This is variable text.

To Specify Entities of a Simple Symbol

To specify entities to be included in the definition of a symbol, you begin by selecting the draft geometry, nodes, unattached notes (defined as variable or fixed text), and filled areas to compose the symbol. Using the commands in the SYMBOL EDIT menu, you can create symbol entities in a number of ways:

* In the symbol edit window, create geometry and text using the **Detail** command.

* In a drawing, create geometry and text; then copy what you need into the symbol edit window using the **Copy Drawing** command.

* Using the **Copy Symbol** command, retrieve a selected symbol into the symbol edit window (for a generic symbol, when adding a symbol, you must select groups to include) and use its entities to define your own symbol. Once the symbol appears in the symbol edit window, the system ungroups its entities, and you can manipulate them individually.

* Import entities into the symbol edit window using the **Import** command in the Pro/ENGINEER **File** menu. You can import IGES, DXF, SET, and CGM drawing files or input from a tablet.

    Notes:

    – You cannot modify cross-hatching in a symbol instance because the symbol is a grouped entity. When you ungroup it, the cross-hatching explodes into individual entities. This behavior also occurs when you are redefining the symbol in the SYMBOL EDIT menu.

    – You cannot include dimensions and coordinate systems in symbols.

Example: Entities Allowed in a Symbol
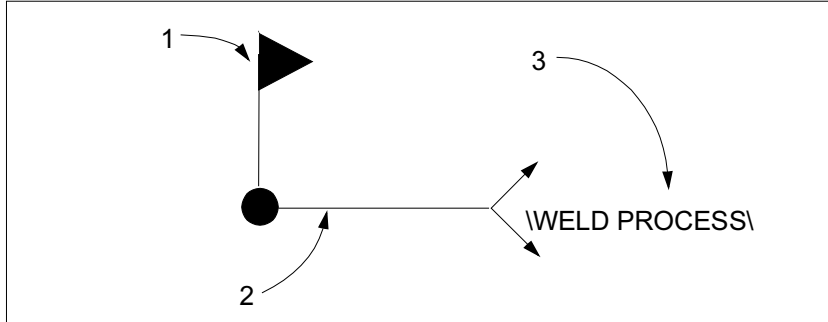
1   Filled area.
2   Draft geometry.
3   Variable text.

Specifying Attributes

Using the Symbol Definition Attributes dialog box, you can specify the following symbol attributes:

- Placement type
  - Free, on an entity, or normal to an entity
  - Left, right, or radial leader
- Symbol instance height
  - Fixed
  - Variable in drawing units, model units, or text-related
- Position and other characteristics
  - At a fixed text angle
  - With an elbow
  - Mirror image of the original geometry or text
- Variable text

To Specify Placement Type of a Simple Symbol

When you choose one of the allowed placement types for a symbol, you must select the origin. You can attach symbols to coordinate systems using leader attachments or using the **On Entity** check box while placing a symbol instance.

Symbol Instance Height

To relate the height of a symbol to a model view that was defined using the **On Item** or **Free Note** placement attributes, select **Variable - Model Units** from the **General** page of the Symbol Definition Attributes dialog box. The system then remembers the height of instances of the symbol in model units. If you change the view scale of the model units, it automatically adjusts the symbol's visible size to maintain a constant proportional relationship with the model.

You can proportion symbols, such as diameter symbols, to the text that follows. In such cases, define the symbol attributes by selecting **Variable - Text Related** from the Symbol Instance dialog box. However, if a symbol (for example, a diameter symbol) does not contain any text to which you can make it proportional, you can create symbol text with a blank line by doing one of the following:

- Create a one-line note in the symbol. From the DRAWING menu, choose **Mod Text** and **Full Note**, and change whatever text you had in the note to a single blank line (make sure to retain the braces and other

special characters). To include this note when defining the symbol, choose **Pick Box** and select **Variable - Text Related** from the Symbol Instance dialog box to define the symbol.

- Include a variable text note when defining the symbol. Select **Variable - Text Related** from the Symbol Instance dialog box to define the symbol. When you create an instance of this symbol, backspace over the default text in the **Var Text** box so that text does not appear in the symbol.

To Relate a Symbol's Height to a Model View

1. Select **Variable - Model Units** from the **General** page of the Symbol Definition Attributes dialog box.
2. Relate the symbol instance to a model. Press the middle mouse button to relate the symbol to the whole drawing, which determines its height in drawing units, or select a specific view.
   Note: Symbols placed on an item or normal to an item are either on an edge of the model and related to that model, or on a draft entity and related to the whole drawing. If a free symbol is located entirely within the view boundary box of a single view, the system automatically selects its view model. Otherwise, you must select a view in which to place the symbol.
3. Specify a value for the symbol instance height in the units of the model.

To Create a Leader with an Elbow

To create a leader with an elbow, select **Allow Elbow** from the **Attributes** box of the Symbol Definition Attributes dialog box. This command is unavailable until you specify a placement type using a leader.

Example: Attaching a Symbol to a Leader with an Elbow
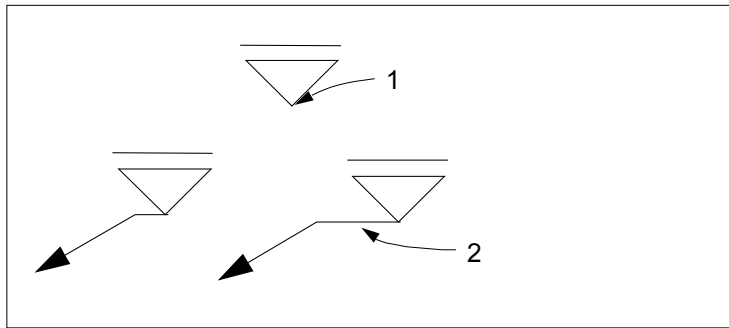
1   Specify the lowest point as the attachment point.
2   You can change the length of the elbow using the **Move Text** command.

To Control Mirror Properties of a Simple Symbol

You can control the way in which a symbol and its notes appear and reorient themselves upon mirroring and rotation by using one of the following three methods:

- Set the drawing setup file option `sym_rotate_note_center` to `yes` (the default value). The system rotates the symbol note as if its origin were in the middle of the height of the text, rather than at the bottom of the text. If you set it to "no," the system rotates the text by rotating its origin point, as it is. Changing the value of this setup option changes the position of existing rotated texts when you repaint.

- Specify attributes in the Symbol Definition Attributes dialog box. You can make a mirror image copy of the original symbol geometry by choosing **Mirror** from the TOOLS menu and selecting a draft line. If you do not select **Geom. Will Mirror** from the Symbol Definition Attributes dialog box, the system mirrors only the symbol origin about the mirror line, and the symbol geometry appears in the same orientation as the symbol you are copying. When you have also selected **Text Will Mirror**, the newly created text is a mirror image of the original text, with the origin translated about the mirror line as well. Thus, the text appears

backward compared to the original text, and its angle and origin also are mirrored.

- Use the Toggle Rotate command in the NOTE ROTATE menu. To rotate individual notes with a symbol, choose **Redefine** from the DWG SYMBOL menu, **Note Rotate** from the SYMBOL EDIT menu, and **Toggle Rotate** from the NOTE ROTATE menu. You can then select a note in the symbol edit window so that it does rotate with the symbol. The **Show Fixed** command highlights notes in the symbol edit window that currently remain fixed in orientation (angle) when their symbol is rotated. This highlighting disappears when you repaint.

To Control Variable Text of a Simple Symbol

Using the **Var Text** page in the Symbol Definition Attributes dialog box, you can specify default values and additional predefined values of variable text for a symbol:

- The specification consists of a list of compound items and an input text area (field).
- Each compound item in the list represents variable text.
- The label to the left of a compound item indicates the name of the variable text.

The system uses the default value (to the left of the variable text name) when it first creates the symbol instance. When you type [&dim] as the default text for symbol definition, you must select a dimension for a particular instance. The value of the dimension disappears from its previous location and appears as the text of the symbol. When you modify variable text, if the default value is &dim, you must select a dimension when you place the symbol.   In addition to the default, you can specify other preset values for each variable text.

You can use variable text to create a unique instance of a symbol by replacing the default text in the symbol with a numeric value, text, or dimension when creating the instance. Pro/ENGINEER adds variable text as an unattached note with the text enclosed in backslashes (\).

Note: Variable text may consist of only one *text* structure. Since Pro/ENGINEER parses special characters as separate texts, you cannot use backslashes to define the value of variable text.

About Defining Generic Symbols

A generic symbol defines a family of similar symbols; it contains all entities pertaining to this family. You can arrange geometry and text in the generic symbol in groups and subgroups, creating a tree structure of symbol definition. Each level of symbol definition, containing more than one group, is characterized by the group attribute restricting the selection of groups at the specified level. That is, you can choose **Exclusive** from the GROUP ATTR menu to define groups, so that you include only one of them in the symbol instance, or you can use the **Independent** command to define them, so that you can select any number of groups (or none). The following figure illustrates how to use groups to assemble symbol instances.

The tree definition structure helps you create symbol instances. When you specify the groups to include, the system assembles an instance out of predefined blocks, or groups. A group can consist of other subgroups and independent entities. The system always selects these independent entities on each level when you select the group to which they belong. Subgroups of different subgroups can share the same name.

Example: Creating a Generic Symbol Definition
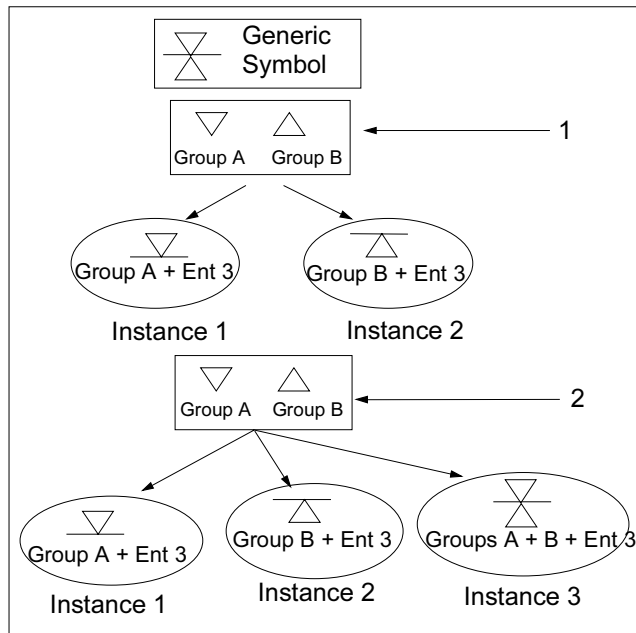
Note: The line is not included in any group because it must appear in all instances.

1   ARRANGEMENT 1: Groups A and B are exclusive.
2   ARRANGEMENT 2: Groups A and B are independent.

Moving Along the Symbol Definition Tree

When defining and redefining symbols, you can move between groups and subgroups using the **Change Level** command in the SYM GROUPS menu. This displays the GROUP menu with the list of existing groups at the current level. Choosing a group from the list moves you down a level so that you can work on the lower level. Choosing **This Level** tells the system that you are on the required level and want to proceed with an action. If you choose **UP**, you move up one level in the symbol definition tree.

To Create a Tree Definition Structure

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **Define**.
3. Type the symbol name, [FILLET].
4. Choose **SYMBOL EDIT > Copy Drawing** to copy entities into the symbol edit window. Choose GET SELECT > **Pick Many** > **Pick Box** to select all entities quickly.
5. Choose **Done Sel** to confirm your selection. All selected entities appear in the symbol edit window.
6. Choose **SYMBOL EDIT > Groups** > **Create**.
7. Type the group name as [ARROW_SIDE]. Select all of the entities that belong to the arrow side except the reference line. The system does not include the reference line in any group because it must appear in all symbol instances.
8. After you select all entities, confirm by choosing **Done Sel**.
9. To create another top-level group, choose **SYM GROUPS > Create**, type the name of the group [OTHER_SIDE], and select entities located on the other side of the reference line. Conclude by choosing **Done Sel**.
10. For this example, top-level groups are exclusive. Choose **SYM GROUPS > Group Attr** > **Exclusive**.
11. To create subgroups of the top-level groups, choose SYM GROUPS > **Change Level**.

12. The TOP LEVEL menu displays the list of groups at the current level and the **This Level** command. Choose **ARROW_SIDE**; all entities pertaining to this group appear in the symbol edit window.
13. Choose **ARROW_SIDE > This Level**.
14. To specify subgroups at the current level, choose **Create**, type the group name, and select the corresponding geometry or text line. When creating subgroups, use the following table:

Group name	What you select on the screen
WELD_SIZE	/weld_size/
LENGTH	/length/
PITCH	/pitch/
CONTOUR	
FINISH	letters C, G, H, M, R, U on top of one another

Note: The system does not include the fillet in any subgroups because it must always be in any arrow-side instance.

By default, the system sets the group attribute to **Independent**; therefore, you do not have to use the GROUP ATTR menu. Groups are independent because you can include all groups in a single instance.

15. To specify the subgroups of the group CONTOUR, choose **Change Level** > **CONTOUR**. The symbol edit window displays entities from the current group.
16. Choose SYM GROUPS > **Create**. Create these three groups:

Group name	What you select on the screen
FLUSH	
CONVEX	
CONCAVE	

17. Choose **SYM GROUPS > Group Attr** > **Exclusive** (you can choose only one of the preceding subgroups under the group CONTOUR at one time).
18. Choose **Change Level** > **UP** > **FINISH**. To select the proper character on the screen, use **Query Sel**. Create groups as follows:

Group name	What you select on the screen
CHIP	C
GRIND	G
MACHINE	M
HAMMER	H
ROLL	R
UNSPECIFIED	U

19. Choose SYM GROUPS > **Group Attr** > **Exclusive**. You have completely described the symbol along the *arrow-side* branch.
20. To specify subgroups along the *other-side* branch, choose **Change Level** > **UP** to return to the TOP LEVEL menu.
21. Choose **OTHER_SIDE** and proceed to create groups, following a procedure similar to the one described in *Steps 8 through 19*.
22. When you finish the symbol definition, choose SYM GROUPS > **Done**.

23. Choose SYMBOL EDIT > **Attributes**.
24. From the Symbol Definition Attributes dialog box, select items characterizing the attachment point, leader type, and symbol size. For this example, click **Left Leader** and **Variable**.
25. Select the leader origin on the left side of the symbol; then click **OK**.
26. Click the **Var Text** tab to type values for notes created between slashes as variable text entries.
27. The system informs you that it has defined the symbol. To save the generic symbol on a disk, choose SYMBOL EDIT > **Write**. Type the directory path or accept the default.

About Adding Nodes

While defining symbols or after defining them, you can add nodes (also referred to as pins) as valid wire attach points on components and connectors. You can also generate, modify, store, and view parameter sets.

Creating Nodes

If you include a node in a symbol, the system identifies it by placing its name in a note. When you create nodes during symbol definition, it automatically adds the PIN command and node name value to the symbol parameter file. The drawing setup file option `node_radius` controls the display of nodes in symbols.

To Create a Node

1. Click **Format > Symbol Gallery >Redefine** to retrieve the symbol to include a node.
2. In the **SYM EDIT** dialog box, click **Insert > Node…**. The **SYMBOL NODE** menu appears.
3. Click **Make Node**.
4. Type the node name in numeric or alphanumeric form.
5. Select the node location on the symbol. A green dot indicating a node with the node name appears on the symbol.

Referencing Parameters in Node Notes

You can edit node notes as text to include the following references to other parameters:

- Node name, for example, `&node_name`
- Node parameters, for example, `&signal_name`
- Top-level symbol parameters, for example, `&ref_des`
- Any valid drawing parameters included as labels, for example `&scale`
  Note: You can reference only those symbols to which you have assigned values in the symbol parameter set. If you delete a parameter that a note references, "***" appears in the note instead of the missing parameter value.

Modifying Parameters in Node Notes

You can modify the `node_name` in a node note only by recreating the node on the symbol using the **Definition** and **Redefine** commands.

In symbol definitions, a node note referencing a parameter has a parameter name displayed in a note while the actual parameter value appears in the message area. Therefore, changing the parameter value is not noticeable in a note; the new parameter value appears only in the message area. When you create a symbol instance, the system replaces parameters included in a node note by their values. Modifying a parameter value updates the node note to show the current parameter value. If you modify symbol or node parameter values in a node note the system updates the values of the corresponding parameters listed in the symbol parameter table.

To Blank Parameters in Node Notes

By default, the system shows node notes when it creates them. To blank all node notes:

1. Click **Format > Symbol Gallery >Redefine** to retrieve the symbol that includes a node.
2. Click **Insert > Node…**. The **SYMBOL NODE** menu appears.
3. Click **Erase Notes**. To redisplay them, choose **Show Notes**.

About Including Symbol Parameters

In Pro/DETAIL, you can include two types of parameters in a symbol:

- Node parameters identifying nodes (only if nodes are present in the particular symbol).
- Symbol definition parameters, or the top parameter set, identifying the symbol. The system replaces them with the corresponding information when you add the symbol to the drawing. Fixed text appears the same for any instance of the symbol. You can use the following parameters in symbol definition:
    - Any of the system parameters for drawings
    - Any user-defined parameters
    - &dwg_name

#–   &!!#–   &-…**Note:** You cannot use a note parameter for the value of variable text.
Depending on the application, a symbol can include the following categories of parameters:

- Required
- Optional, system-defined
- Optional, user-defined

If an application uses a symbol with required parameters, but they are missing, the system issues an error message and you can edit the symbol parameter file. The next figure shows the format of the parameter file.

Parameter Set Format

	Symbolparameters	Values
Top parameter set	TYPE	COMPONENT or CONNECTOR <read-only>
	REF_DES	instance reference name <text>
	MODEL NAME	name of reference solid model <text>
	NUM_OF_PINS	number of nodes <integer>
	GENDER	MALE or FEMAL <read-only>
	Node name	Node parameter SIGNAL_NAME	Node parameter SIGNAL_VALUE
List of nodes	PIN Value PIN	Value	Value

Pro/ENGINEER uses symbols containing an individual parameter set to create electrical diagrams. You can define Pro/DIAGRAM symbols in Drawing mode and Diagram mode. However, if you want to use them in Pro/DIAGRAM, you must define them as components or connectors by providing the parameter set appropriate for the type of object the symbol represents (component or connector). The following tables list component and connector parameters (the required parameters are shown in bold type).

Component Parameters

PARAMETER	DESCRIPTION
DESCRIPTION	The description of the component (such as FUEL GAUGE).
TYPE	The object type (COMPONENT).

REF_DES	The reference designator name. You must store the default name with the symbol definition. When placing a component symbol, you must supply a unique REF_DES name to be stored with the symbol instance and used in wire lists.
MODEL_NAME	The name of the reference solid model, that is, the physical model represented by the component symbol.
NUM_OF_PINS	The number of logical pins. The system determines the number of visible pins by the number of nodes in the symbol definition.
PIN	Pin names and signals information. For each node included in the symbol definition, the system generates a line in the following format: PIN PIN_NAME SIGNAL_NAME SIGNAL_VALUE ENTRY_PORT PIN_NAME is supplied automatically; SIGNAL_NAME and SIGNAL_VALUE are optional; you can type them manually.

Connector Parameters

PARAMETER	DESCRIPTION
DESCRIPTION	The description of the connector (such as 12 PIN FEMALE).
TYPE	The object type (CONNECTOR).
REF_DES	The reference designator name.
MODEL_NAME	The name of the reference solid model, that is, the physical model represented by the connector symbol.
GENDER	MALE or FEMALE.
PIN	Pin names and signals information. For each node included in the symbol definition, the system generates a line in the following format: PIN PIN_NAME SIGNAL_NAME SIGNAL_VALUE ENTRY_PORT PIN_NAME is supplied automatically; SIGNAL_NAME and SIGNAL_VALUE are optional; you can type them manually.
NUM_OF_PINS	Number of logical pins. Number of visible pins for fixed connectors is defined by the number of nodes in the symbol definition, for parametric connectors, by the Num Vis Pins attribute value.

You can perform the following procedures on parameters:

- Generate, edit, or view a symbol definition parameter set.
- Store a symbol definition parameter set in a file.

To Generate a Parameter Set

To create a parameter set, you can do one of the following:

- Generate a set of default parameters by reading in data (system- or user-defined).
- Type parameter names and their values in a table using the Pro/TABLE environment.

To Generate a Set of Default Parameters

You can generate a set of default parameters to symbols.
1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **DWG SYMBOL > Refine** to locate the symbol to assign default parameters.
3. Click **SYMBOL EDIT > Parameters** > **Read**.
4. Using the READ SYM PRM menu, do one of the following:
   - Choose **Comp Default** and retrieve a system set of default parameters for components.
   - Choose **Conn Default** and retrieve a system set of default parameters for connectors.

- Choose **Other** and retrieve a user-specified file (with the extension ".spm") containing the appropriate parameters.

To Create Parameters in Pro/TABLE

5. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
6. Click **DWG SYMBOL > Refine** to locate the symbol to assign default parameters.
7. Click **SYS PARAMS > Modify.** A blank Pro/TABLE window appears unless you read in a parameter file previously.
8. Using the Pro/TABLE editor, type parameters and their values in a required format. Use the Pro/TABLE **Help** command or **Edit**. Choose **Keywords** (equivalent key is F4) when needed. You can edit the parameter set later as needed.

System Parameters for Drawings

The following table lists all system parameters available for use in drawings, classified according to functionality.

PARAMETER NAME	DEFINITION
&d#	Displays a dimension in a drawing note, where # is the dimension ID.
&ad#	Displays an associative dimension in a drawing note, where # is the dimension ID.
&rd#	Displays a reference dimension in a drawing note, where # is the dimension ID.
&p#	Displays an instance number of a pattern in a drawing note, where # is the pattern ID.
&g#	Displays a gtol in a drawing note, where # is the gtol ID.
&<param_name>	Displays a user-defined parameter value in a drawing note.
&<param_name>:att_cmp	An object parameter that indicates the parameters of the component to which a note is attached.
&<param_name>:att_edge	An object parameter that indicates the parameters of the edge to which a note is attached.
&<param_name>:att_feat	An object parameter that indicates the parameters of the feature to which a note is attached.
&<param_name>:att_mdl	An object parameter that indicates the parameters of the model to which a note is attached.
&<param_name>:att_pipe_bend	An object parameter that indicates the parameters of the pipe bend to which a note is attached.
&<param_name>:att_spool	An object parameter that indicates the parameters of the spool to which a note is attached.

&<param_name>:EID_<edge_name>	An object parameter that references edges.
&<param_name>:FID_<feat_ID>	An object parameter that includes a feature parameter in a note by ID.
&<param_name>:FID_<FEAT_NAME>	An object parameter that includes a feature parameter in a note by name.
&<param_name>:SID_<surface_name>	An object parameter that references surfaces.
&angular_tol_0_0	Specifies the format of angular tolerance values in a note from one to six decimal places.
&current_sheet	Displays a drawing label indicating the current sheet number.
&det_scale	Displays a drawing label indicating the scale of a detailed view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates this parameter with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&dtm_name	Displays datum names in a drawing note, where name is the name of a datum plane. The datum name in the note is read-only, so you cannot modify it; unlike dimensions, a datum name does not disappear from the model view if included in a note. The system encloses its name in a rectangle, as if it were a set datum.
&dwg_name	Displays a drawing label indicating the name of the drawing.
&format	Displays a drawing label indicating the format size (for example, A1, A0, A, B, and so forth).
&linear_tol_0_0	Specifies the format of dimensional tolerance values in a note from one to six decimal places.
&model_name	Displays a drawing label indicating the name of the model used for the drawing.
&parameter:d	Adds drawing parameters to a drawing note, where *parameter* is the parameter name and *:d* refers to the drawing. .
&pdmdb	Displays the database of origin of the model.
&pdmrev	Displays the model revision.
&pdmrev:d	Displays the revision number of the model (where *:d* refers to the drawing).
&pdmrl	Displays the release level of the model.
&scale	Displays a drawing label indicating the

	scale of the drawing.
&scale_of_view_detailed_bar	
&sym(`<symbolname>`)	Includes a drawing symbol in a note, where `symbolname` is the name of the symbol.
&todays_date	Displays a drawing label indicating the date on which the note was created in the form dd-mm-yy (for example, 2-Jan-92). You can edit it as any other nonparametric note, using **Text Line** or **Full Note**.
	If you include this symbol in a format table, the system evaluates it when it copies the format into the drawing.
	To specify the initial display of the date in a drawing, use the configuration file option "todays_date_note_format."
&total_sheets	Displays a drawing label indicating the total number of sheets in the drawing.
&type	Displays a drawing label indicating the drawing model type (for example, part, assembly, etc.).
&view_name	Displays a drawing label indicating the name of the view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&view_scale	Displays a drawing label indicating the name of a general scaled view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.

Pro/REPORT System Parameters

&asm.mbr.comp....	Retrieves information about the component from the model data and displays it in the report table.
&asm.mbr.cparam....	Retrieves a given component parameter.
&asm.mbr.cparams....	Lists information pertaining to all component parameters for the current model.
&asm.mbr.name	Displays the name of an assembly

	member. To show tie wraps and markers, the region attributes must be set to **Cable Info**.
&asm.mbr.param....	Displays information about parameters in an assembly member.
&asm.mbr.type	Displays the type (part or assembly) of an assembly member.
&asm.mbr.User Defined	Lists the specified user-defined parameter for the respective assembly components. Note that "&asm.mbr." can be used as a prefix before any user-defined parameter in an assembly member.
&dgm....	
&fam....	Retrieves family table information about the model.
&harn....	Shows cable harness parameters for 3-D harness parts and flat harness assemblies.
&lay....	Retrieves layout information about the model.
&mbr....	Retrieves parameters about a single component.
&mdl....	Retrieves information about a single model.
&prs....	Retrieves process-specific report parameters used to create reports on the entire process sequence.
&rpt....	Displays information about each record in a repeat region.
&weldasm....	Retrieves welding information about the model.
&asm.mbr.cblprm....	Lists values for a given cabling parameters.
&asm.mbr.cblprms....	Lists values for cabling and wire parameters.
&asm.mbr.connprm....	Lists parameters for connector pins in flat harness assemblies.
&asm.mbr.pipe....	Shows pipeline, pipe segment, and Pro/REPORT bend information parameters.
&asm.mbr.generic.name….	Lists the generic name information for a family table instance in a table.
&asm.mbr.topgeneric.name….	Lists the top generic name information for a family table instance in a table when working with a nested family table.

To Edit Symbol Parameters

Once you have created a parameter set, you can modify symbol and node parameters by editing the parameter file using the Pro/TABLE editor. You can change the parameter value, add user-defined parameters, or delete parameters. If the system encounters errors after you have edited the parameter file, it displays an Information Window indicating the type of errors. You can use commands in the REEDIT menu to resolve the discrepancies.

To View Symbol Parameters

Using the **Show** command in the SYM PARAMS menu, you can view the symbol definition parameter set.

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **DWG SYMBOL > Refine** to locate the symbol to view parameters.
3. Click **SYMBOL EDIT > Parameters > Show.**

Storing Symbol Parameters

While working with a symbol definition, you can save the parameter set to a file in your working directory, and read it in later to create similar parameter sets for other symbol definitions.

To Store a Parameter File

1. Click **Format > Symbol Gallery**. The **DWG SYMBOL** menu appears.
2. Click **DWG SYMBOL > Refine** to locate the symbol to store parameters.
3. Click **SYMBOL EDIT > Parameters > Write.**
4. Type the filename without an extension. The system stores the file with the name "filename.spm" to disk.
5. If the name that you typed for the parameter file already exists in your working directory, type [Y] to replace the existing file or [N] to avoid overwriting; then type a new filename.

About Adding Symbols to a Drawing

The Symbol Instance dialog box is used to transfer symbols to and from the drawing and symbol instance pallette. When you add a symbol to a drawing, you create an instance. To find the name and directory path of the symbol corresponding to an instance, use the **Show Name** command in the **DWG SYMBOL** menu. Using the Symbol Instance dialog box, you can create new symbol instances and preview them during the creation process. Specifically, you can do the following:

- Specify placement characteristics:
  - Content
  - Height
  - Angle
  - Position
- Specify the groups to include in the symbol.
- Modify variable text.
- Adjust instance location.

Symbol instances exist *only* in the drawing format. In Drawing mode, you cannot edit symbols that you created and added to formats in Format mode if you add the format to a drawing. To edit the symbol, retrieve the format in Format mode and make any of the necessary changes. To access format symbols in Drawing mode, write the symbol to disk and retrieve it into the drawing from the appropriate directory in which the file is stored.

To Place a Symbol Instance

1. Click **Insert > Symbol Instance**. The **Symbol Instance** dialog box opens.
2. In the **Symbol Instance** dialog box, choose one of the following options:

- **Pick in Draw** — Select a symbol in the drawing.
- **Retrieve** — Retrieve a system symbol from disk.
- **Sym Palette** — Select a symbol from another drawing.

Note: In order to use **Sym Palette**, the configuration option linking to the drawing containing the symbol must be established.

3. Specify the height of the instance by typing a value in the **Height** box.
4. Specify the placement type by selecting an item from the **Type** list.
5. Click **Place Inst...** to place the symbol in the drawing.
   - If you chose alternative specifications in the Symbol Definition Attributes dialog box (such as **On Entity** and **Free**) before adding an instance to the drawing, indicate how to place the instance by using commands in the **INST ATTACH** menu.
   - If you define a symbol with a leader command, specify the leader attachment type by choosing a command from the **ATTACH TYPE** menu.
6. The system displays the symbol as specified.

To Create a Symbol Instance

1. Click **Insert > Symbol Instance**. The **Symbol Instance** dialog box opens
2. Click **Pick in Draw** to select a symbol in the drawing, **Retrieve** to retrieve a system symbol from disk, or **Pick in Palette** to use the symbol palette.
3. Type the height and value of the instance.
4. Specify the placement type by selecting an item from the **Type** list.
5. Click **Place Inst**… to place the symbol in the drawing.
   - If you choose alternative specifications in the Symbol Definition Attributes dialog box (such as **On Entity** and **Free**) before adding an instance to the drawing, indicate how to place the instance by using commands in the **INST ATTACH** menu.
   - If you define a symbol with a leader command, specify the leader attachment type by choosing a command from the **ATTACH TYPE** menu.

To Repeat a Symbol Instance

1. In the **Symbol Instance** dialog box, click **Pick in Draw** to select a symbol in the drawing, click **Retrieve** to retrieve a system symbol from disk, or **Pick in Palette** to use the symbol palette.
2. Click **Place Inst**, and then select a location to place the symbol instance.
3. In the **Symbol Instance** dialog box, click **Repeat Inst**. This action creates a copy of the last symbol instance you selected.
4. Select a location to place the symbol instance.

Example: Symbol with Leader
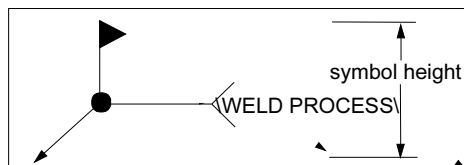
To Relate a Symbol Instance to Dimension Text

Before or after you create a symbol, you can relate it directly to dimension text so that it moves with the dimension when the dimension changes location. When specifying the placement of a new symbol instance, select **Offset** from the **Placement Type** list in the Symbol Instance dialog box. Then select dimension text and a location relative to the text. To relate an existing symbol to dimension text, choose **Relate Obj** from the TOOLS menu.

To Create an Offset Attachment to a Symbol or Note

You can attach a symbol or note that is offset from a dimension leader arrow endpoint.

1. Click **Format > Symbol Gallery >Redefine** to retrieve the symbol to create an offset attachment.
2. In the **SYM EDIT** dialog box, click **Insert > Note**. The **SYMBOL NODE** menu appears.
3. Click **Insert > Note > Make Note**.
4. Select one of the following:
   - Dimension
   - Dimension arrow
   - Note
   - Balloon
   - GTOL
   - Symbol instance
   - Reference dimension
5. Select the location for the symbol or note, and then select a symbol from the **Symbol Palette** dialog box or type the note in the text box.

Creating an Offset Attachment to a Symbol or Note

Symbols and notes can be attached with an offset to other symbols and notes. When the parent symbol or note is moved, the child symbol or note moves with it and maintains the offset distance. You can also attach a symbol or note that is offset from a leader arrow endpoint.

Specifying the Grouping of a Symbol Instance

Using the **Grouping** page of the Symbol Instance dialog box, you can display the tree representation of the grouping hierarchy of the symbol.

To expand and contract tree elements, select the square to the left of the subgroup name as follows:

- Select the subgroup once to include it in the instance.
- Select it again to remove it (and all subgroups) from the symbol.

Enabling or disabling a group does not expand or contract its subgroups; however, you can select the square to the left of the subgroup name to modify the tree view. If you include a subgroup, this enables all parents recursively (if you have not already enabled them). If you disable a parent, you disable all children. If you re-enable a parent, the system restores the status of its subgroups.

To Create an Instance of a Generic Symbol

1. Click **Insert > Symbol Instance**.
2. In the Symbol Instance dialog box, click **Pick in Draw** to select a symbol in the drawing, or click **Retrieve** to retrieve a system symbol from disk.
3. Specify the height of the instance by typing a value in the **Height** box.
4. Specify the placement type by selecting an item from the **Type** list.
5. Click **Grouping** to specify the groups and subgroups to include in the particular instance. Only the groups that you select from group namelist menus appear in the instance.
6. The dialog box displays a tree representation of the grouping hierarchy of the symbol. Select groups as follows:
   - If groups are independent, select any number of groups.
   - If groups are exclusive, select only *one* group to include in the symbol.
7. Expand the tree structure to show the subgroups, and select the ones that you want. Click **OK**.

Modifying Variable Text in an Instance

Using the **Var Text** page of the Symbol Instance dialog box, you can change the content of a note included in a symbol instance if the text was created as variable text.

You can modify text parameters when redefining the symbol. However, when you attempt to modify text parameters in instances, you should consider the following:

- For variable-height symbols, you cannot modify text height independently of the symbol size.
- For fixed-height symbols, you can modify text only when you are redefining a symbol. Choose **Mod Text** from the SYMBOL EDIT menu to access text editing commands.

To Modify the Values of Variable Text

1. Click **Insert > Symbol Instance**.
2. Select the symbol instance to modify.
3. In the Symbol Instance dialog box, click **Var Text**. The dialog box displays a set of predefined values for the symbol instance from which you can choose only one. The predefined values, like the default values of variable texts, are part of the symbol definition.
4. Specify default values and additional predefined values of variable text, or select variable text of the instance in the drawing in the preview window and then modify it.

To Adjust Instance Location

You can change the position of a symbol instance any time after you add it to the drawing. Using the **Move**, **Move Text**, or **Mod Attach** buttons in the Symbol Instance dialog box, you can adjust the location of an instance by dragging it. As you place a symbol, it moves dynamically, attached to the cursor. When you place it, the fixed symbol text appears.

- If the symbol has variable text, type the text value.
- If you are relocating a symbol and decide not to place it dynamically, press the middle mouse button to exit (this functionality is not available for **Variable - Model Units** symbols).

About Manipulating Instances

Using the Pro/DETAIL menu, you can modify, display, erase, and delete instances; include instances in notes; and relate instances to dimension text. You can also show symbols for welds that you create in Pro/WELDING.

Modifying Instances

After you create a symbol, you can use the Symbol Instance dialog box to modify it in the following ways:

- Explode (snapshot) a symbol instance.
- Change the symbol height.
- Change the rotation angle.
- Move the origin of the symbol.
- Change the grouping attributes.
- Modify variable text.

You can also reattach instances and add new leader lines.

To Modify a Symbol Instance

1. Click **Insert > Symbol Instance**. The **Symbol Instance** dialog box opens.
2. Select an instance.
3. Using the Symbol Instance dialog box, modify the instance by doing one or all of the following:

- Explode the instance by selecting **Explode** from the **Symbol Definition Usage** list. Pro/ENGINEER copies the entities of the selected instance to a drawing, and deletes the instance. A node is represented by a filled dot, and the note that the system creates by exploding the node note is attached to the dot. It only copies visible symbol entities, and deletes the symbol leader (if it exists).
- Change the symbol height by typing a value in the **Height** box.
- Change the rotation angle by typing a value in the **Angle** box.

Note: Before rotating a symbol instance, you can reset the parameter that controls text positioning by redefining the symbol definition using the **Attributes** command in the SYMBOL EDIT menu; however, modifying symbol attributes affects *all* instances of this symbol definition.

- Move the origin by clicking **Move Origin** and selecting a view.

4. When you have finished, click **OK** to update the display of the instance.

To Rotate Symbols with Text

When you are rotating a symbol, any text included in a symbol behaves according to how you defined it in symbol attributes (that is, whether you selected the **Fixed Text Angle** check box in the **Attributes** box of the Symbol Definition Attributes dialog box).

Using the drawing setup file option sym_flip_rotated_text, you can flip the text of a rotated symbol. If you set it to yes (the default is no), and the symbol orientation is +/- 90 degrees, all of the text that you defined in the **Attributes** box of the Symbol Definition Attributes dialog box menu flips, rotating along with the symbol.

Examples: Rotating Symbols with Text

Rotating a Symbol with Text
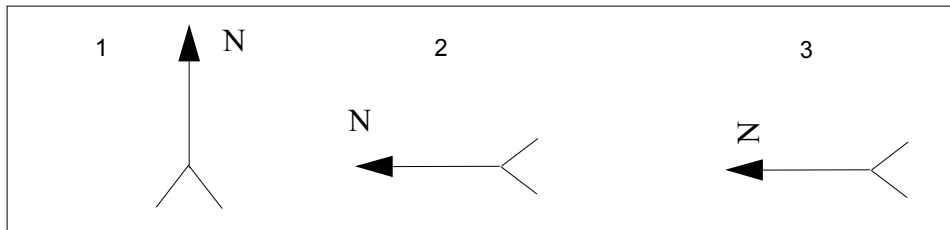
1 The symbol in its original position.
2 The symbol created when using the **Fixed Text Angle** check box.
3 The symbol created when using **Fixed Text Angle** and rotating the symbol during instance creation.

Flipping Text Using "sym_flip_rotated_text"
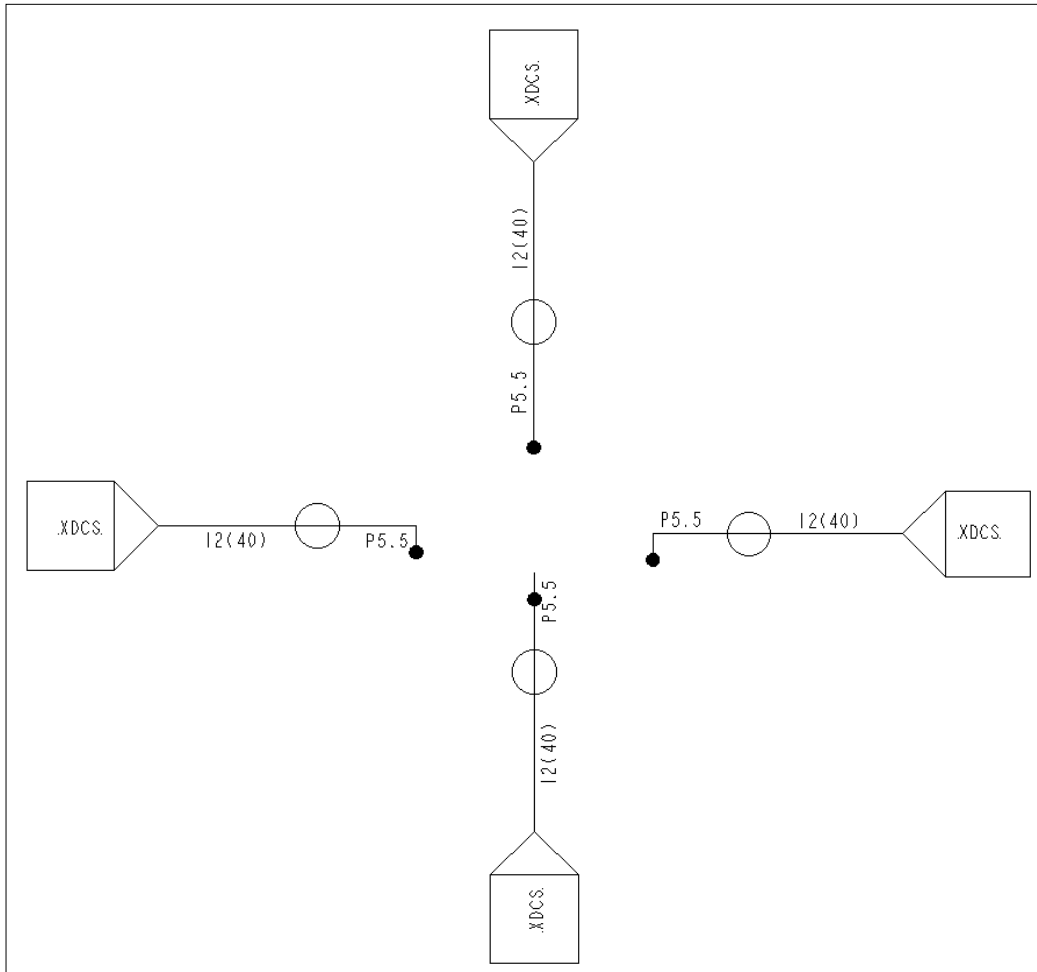
To Reattach a Symbol Instance

Using the **Mod Attach** command in the Pro/DETAIL menu, you can reattach a symbol instance with a leader any time after you have added it to a drawing by doing one of the following:

- Reattach the symbol leader to a new point on geometry in a drawing.
- Reattach a symbol leader to a new point on an entity that includes a symbol.

Adding New Leader Lines

You can add leaders to symbols (and delete them) following the same procedure that you use to add leaders to drawing notes. However, the following restrictions apply:

- You cannot add new leaders to BOM symbols or to datum target symbols.
- To add a leader to a user-defined symbol, you must specify an attribute in the symbol definition to place the symbol with a leader. Similarly, to delete a leader from a user-defined symbol, you must specify an attribute in the symbol definition to place the symbol without the leader.

To Add Another Leader Line to a Symbol

1. Click **Edit > Attachment**.
2. Select a reference point on the view for the leader.
3. Click **Add Ref**.
4. Choose **GET SELECT > Done Sel** and **ATTACH TYPE > Done**. The system adds the additional leader.
   - To add more than one leader, repeat Steps 2 and 3.
   - To delete a leader, choose **Delete Ref**.

To Erase and Display Symbol Instances

Using the **Show and Erase** dialog box, you can erase symbol instances from a drawing by blanking selected symbols. To redisplay them, click **Show**.

To Delete Symbol Instances

You can remove symbol instances from a drawing by using **Delete** in the Pro/DETAIL menu. When you delete an instance from the drawing, the system does not delete the symbol definition from the drawing. To remove it, choose **Delete** from the DWG SYMBOL menu.

About Working with Parametric Weld Symbols

You can show weld symbols in drawings for welds that were created in Pro/WELDING, as well as redefine the system-supplied parametric weld symbols to improve your flexibility and productivity.

Showing Symbols for Welds Created in Pro/WELDING

You can show and erase weld symbols in drawings that correspond to welds that were created in an assembly using Pro/WELDING. You can also do the following:

- Modify the number of decimal places (*num digits*) shown in dimensions contained within a weld feature.
- Add or delete leader lines to weld symbols.

To display weld symbols from Pro/WELDING in drawings according to the ANSI or ISO standard, set the value of the drawing setup file option `weld_symbol_standard` to `STD_ANSI` or `STD_ISO`, respectively.

Restrictions on the Use of Weld Symbols in Drawings

When you are working with weld symbols, the following restrictions apply:

- The display of the weld feature does not affect the display of the weld symbol, and it does not update its point of attachment when you blank, resume, erase, or show the weld feature. However, when the system first shows a weld symbol, its default attachment adapts to the display status of its feature.
- The system shows a weld symbol only once in a drawing, similar to assembly geometric tolerances and surface finishes.
- Revision 15.0 and later revisions support the following compound welding symbols:
  - Reinforced Welds: square groove, bevel groove, flared bevel groove, and J-groove (all reinforced welds can also be two-sided).
  - Two-sided Welds: fillet, square groove, V-groove, bevel groove, U-groove, J-groove, flared V-groove, flared bevel groove.

To Show Weld Symbols in a Drawing

1. Choose **View > Show and Erase**.
2. In the **Show and Erase** dialog box, click **Show** and select **Symbol** from the **Type** box.
3. Select an item from the **Show By** box to specify a feature, view, or part in which to show the weld symbols.

To Erase Weld Symbols

1. Choose **View > Show and Erase**.
2. In the **Show and Erase** dialog box, click **Erase** and select **Symbol** from the **Type** box.
3. Choose the appropriate buttons to select weld features, and specify the appropriate weld symbols. The system erases the specified symbols.

To Show or Erase Weld Features in a Drawing

1. Choose **View > Show and Erase**.
2. In the **Show and Erase** dialog box, click **Show** or **Erase**; then select **Cosmetic Feature** from the **Type** box.
3. Choose the appropriate option buttons and commands to show weld features in the drawing or erase them.

Modifying the Number of Decimal Places of a Fillet Weld Feature

When you are using **Num Digits** to modify a value of a fillet weld feature (simple, two-sided, or reinforcing a groove) that has differing leg length values shown in the symbol, such as "L1 x L2," the number of decimal places shown in the two values are linked. That is, if you select this portion of the symbol for modification, the system highlights the entire "L1 x L2," and both values change to the number of decimal places you specify.

To Regroup Weld Symbol Instances

To regroup weld symbol instances, set the configuration file option `sym_leader_orient_move_text` to `yes` (the default is `no`). The system then regroups an instance after you move the text.

User-Defined Parametric Weld Symbols

You can replace the Pro/ENGINEER-supplied library of system weld symbols with user-defined ones. After you define the symbols, the system uses them for automatic weld annotation. By customizing your weld symbols in advance, you can increase your flexibility and productivity throughout the processes of creating and modifying drawings.

Note: Before performing this procedure, you should copy the system-supplied welding symbol library to a backup directory.
However, when creating user-defined weld symbols, the following restrictions apply:

- All of the groups that existed in the original definition must remain in the new definition, and you cannot add new ones or change the names of existing ones.
- If you add new variable text, or change the name of an existing piece of variable text, the new name must be the same as that of the existing variable text in the original.
- The height type of the symbol instance must be the same in the new user-defined symbol as it was in the original.
- The **Left Leader** and **Right Leader** placement types must both exist in the new user-defined weld symbol.

ISO Welding Symbols

This section presents examples of symbols that enable you to create welding, brazing, and examination symbols in a drawing according to the ISO standard. To help you create symbol instances, it provides the following information for each symbol:

- The symbol name
- An example of an instance created from this symbol and the associated menu picks
- Listings of groups and subgroups in menu format reflecting a tree symbol definition structure

You can also refer to **Glossary of Menu Picks** below for an explanation of menu selections for welding symbols.

Welding Symbols Library

The Welding Symbols Library, available with Pro/DETAIL, provides a collection of symbols based on the standards of ISO-2553-1984. Using this library, you can create a variety of welding, brazing, and examination symbols in a drawing.

To retrieve a symbol from the Welding Symbol Library, choose System Syms from the Select File menu to access the Pro/ENGINEER symbols area.

Glossary of Menu Picks

This glossary describes terms that are unique to the ISO Welding Symbol Library and identifies information you are prompted to enter. Menu pick names track the ISO Welding Symbol Library to the fullest extent possible. (For terms that are also used in the ANSI Welding Symbol Library—and are, therefore, not unique to the ISO Welding Symbol Library—see Glossary of Menu Picks in Appendix B, ANSI Generic Welding Symbols).

In the ISO Welding Symbol Library, two lines form the reference line, one solid and one dashed. Weld symbols attached to the solid line are "arrow-side." Weld symbols attached to the dashed line are "other-side." The dashed line is only omitted for symmetrical welds. When creating a welding symbol, you first select whether the weld is arrow side, other-side, or symmetrical. If you choose ARROW_SIDE or OTHER_SIDE, the system prompts you to create the weld symbol above or beneath the reference line. It then places the dashed line appropriately in the symbol depending on whether it is arrow-side or other-side.

Table 1: ISO Weld Characteristics Menu Selections

MENU PICK	OPTION	DESCRIPTION
ABOVE_REF		Creates the weld symbol above the solid reference line.
AS_ABOVE_ REF		Indicates (for combination welding symbols only) whether the arrow-side welding symbol is placed above or beneath the reference line.
BENEATH_REF		Creates the weld symbol beneath the solid reference line.
BUTT_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
CONTOUR	SMOOTH_BLEND	Describes the final shape of the weld. This choice indicates that the toes of the weld are blended smoothly.
FINISH		In the ANSI standard, this specifies the method of finishing. In the ISO standard, the system adds a machine finish surface texture symbol above the contour symbol.
FILLET_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or

		"OS" (other-side) to indicate the appropriate weld side.
FIL_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
HOLE_DIA		Refers to the diameter of a plug weld.
OS_ABOVE_ REF		Indicates (for combination welding symbols only) whether the other-side welding symbol is placed above or beneath the reference line.
PROJECT_DIA		Refers to the diameter of a projection weld.
SEAM_WIDTH		Refers to the width of a seam weld.
SPOT_DIA		Refers to the diameter of a spot or fusion weld.
STYLE		Refers to the longitudinal dimension of a weld. Two choices are available:
	CONTINOUS	Weld joint continuously for the specified length.
	INTERMITTENT	Creates weld beads of specified length, distance between adjacent weld elements, and specific number of beads.
SYMMETRICAL		Refers to a welding symbol for creating identical welds on both arrow-side and other- side.
TAIL		Creates a tail on the opposite end of the welding symbol from the leader attachment. Provides two choices:
	REFERENCE WELD_PROCESS	Enters reference text or note.
		Specifies a welding process.
WELD_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
WELD_TYPE		For combination welding symbols, indicates the type of weld being created and whether it is created arrow-side or other-side (e.g., FILLET_AS).

Bevel Butt Symbol: Bevel_Butt.sym

EXAMPLE 1

PICKS

1. arrow_side
2. above_ref
3. style
4. contour
5. back_type
6. finish
7. leader_orient
8. intermittent
9. convex

10. strip
11. left
12. removable
13. field
14. all_around
15. tail
16. weld_process

EXAMPLE 2

PICKS

1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size

11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference

EXAMPLE 3

PICKS

1. symmetrical   11. convex
2. style_as   12. flat
3. style_os   13. left
4. contour_as   14. all_around
5. contour_os   15. field
6. finish_as
7. finish_os
8. leader_orient
9. intermittent
10. continuous

8x30(60)
300

Symbol Definition Structure

```
                                        ┌─────────────────────┐
                                        │   BEVEL_BUTT.SYM     │
                                        └─────────────────────┘
                                                  │
                                                  ▼
                    ┌──────────────┐     ┌──────────────────┐
                    │ ABOVE_REF    │◄────│ ARROW_SIDE       │
                    │ BENEATH_REF  │     │ OTHER_SIDE       │
                    └──────────────┘     │ SYMMETRICAL      │
                           │             └──────────────────┘
                           ▼                      │
      ┌──────────────┐  ┌──────────────────┐      │
      │ CONTINUOUS   │  │ LEADER_ORIENT    │      │
      │ INTERMITTENT │  │ STYLE            │      │
      └──────────────┘  │ CONTOUR          │      │
             ▲ │        │ BACK_TYPE        │      ▼
             │ ▼        │ FINISH           │   ┌──────────────────┐
      ┌──────────────┐  └──────────────────┘   │ STYLE_AS         │
      │ FLAT         │                          │ STYLE_OS         │
      │ CONVEX       │       ┌──────────────┐   │ CONTOUR_AS       │
      │ SMOOTH_BLEND │       │ CONTINUOUS   │◄──│ CONTOUR_OS       │
      └──────────────┘       │ INTERMITTENT │   │ FINISH_AS        │
 ┌──────────────┐            └──────────────┘   │ FINISH_OS        │
 │ CONTINUOUS   │                               │ LEADER_ORIENT    │
 │ INTERMITTENT │                               └──────────────────┘
 └──────────────┘                                        │
        ▲                                                ▼
        │         ┌──────────────┐                ┌──────────────┐
        │         │ BACK_SIZE    │                │ LEFT         │
        │         │ STYLE        │                │ RIGHT        │
        │         │ CONTOUR      │                └──────────────┘
        │         │ FINISH       │                       │
        │         └──────────────┘                       ▼
 ┌──────────────┐       │        ┌──────────────┐  ┌──────────────┐
 │ FLAT         │   ┌───────────┐│ LEFT         │  │ ALL_AROUND   │
 │ CONVEX       │   │ BACKING   ││ RIGHT        │  │ FIELD        │
 │ SMOOTH_BLEND │   │ STRIP     │└──────────────┘  │ TAIL         │
 └──────────────┘   └───────────┘       │          └──────────────┘
        │                 │             │                 │
        ▼                 │             ▼                 ▼
 ┌──────────────┐         │      ┌──────────────┐  ┌──────────────┐
 │ PERMANENT    │         │      │ FLAT         │  │ REFERENCE    │
 │ REMOVABLE    │         │      │ CONVEX       │  │ WELD_PROCESS │
 └──────────────┘         │      │ SMOOTH_BLEND │  └──────────────┘
                          ▼      └──────────────┘
                   ┌──────────────┐
                   │ ALL_AROUND   │
                   │ FIELD        │
                   │ TAIL         │
                   └──────────────┘
                          │
                          ▼
                   ┌──────────────┐
                   │ REFERENCE    │
                   │ WELD_PROCESS │
                   └──────────────┘
```

Inclined Joint Symbol: Inclined_Joint.sym

EXAMPLE 1

PICKS

```
 1. arrow_side
 2. weld_size
 3. finish
 4. flat
 5. leader_orient
 6. left
 7. field
 8. all_around
 9. tail
10. weld_process
```


EXAMPLE 2

PICKS

```
1. other_side
2. weld-size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference
```

Symbol Definition Structure

```
                         ┌─────────────────────┐
                         │ INCLINED_JOINT.SYM  │
                         └─────────────────────┘
                                    │
                                    ▼
                         ┌─────────────────────┐
                      ┌──│ ARROW_SIDE          │
                      │  │ OTHER_SIDE          │
                      │  └─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │ WELD_SIZE           │
            │ FINISH              │
            │ FLAT                │
            │ LEADER_ORIENT       │
            └─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │ LEFT                │
            │ RIGHT               │
            └─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │ ALL_AROUND          │
            │ FIELD               │
            │ TAIL                │
            └─────────────────────┘
                  │
                  ▼
      ┌─────────────────────┐
      │ REFERENCE           │
      │ WELD_PROCESS        │
      └─────────────────────┘
```
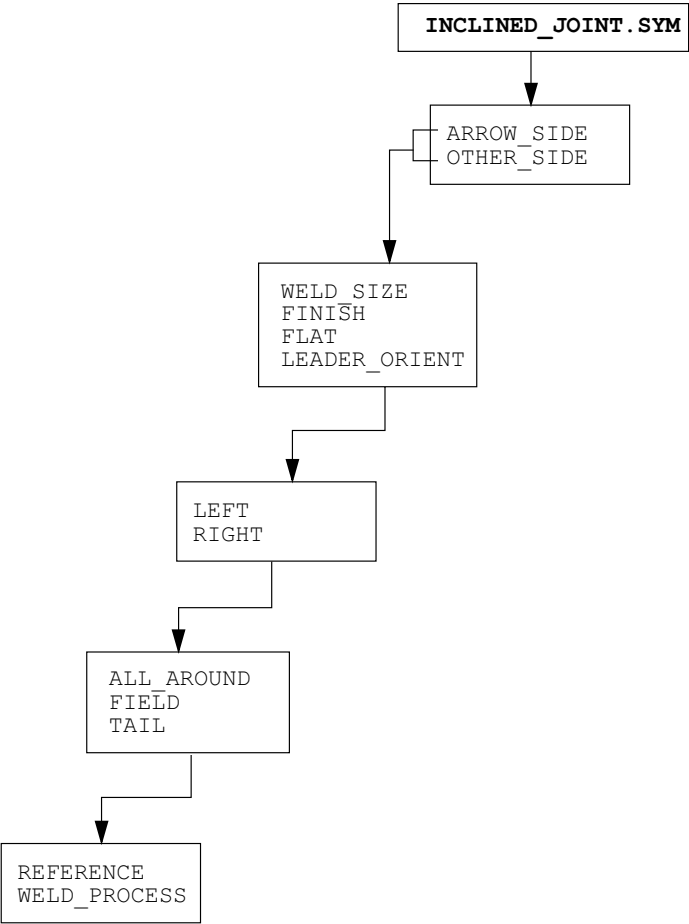
Bevel Butt with Broad Root Face: Br_Root_Bevel.sym

EXAMPLE 1

PICKS

1. arrow_side	11. strip
2. above_ref	12. left
3. weld_size	13. removable
4. style	14. field
5. contour	15. all_around
6. back_type	16. tail
7. finish	17. weld_process
8. leader_orient	
9. intermittent	
10. convex	

EXAMPLE 2

PICKS

1. other_side	11. style
2. beneath_ref	12. contour
3. style	13. finish
4. contour	14. intermittent
5. back_type	15. smooth_blend
6. leader_orient	16. left
7. continuous	17. field
8. flat	18. all_around
9. backing	19. tail
10. back_size	20. reference

EXAMPLE 3

PICKS

1. symmetrical   11. intermittent
2. weld_size_as   12. continuous
3. weld_size_os   13. convex
4. style_as   14. flat
5. style_os   15. left
6. contour_as   16. all_around
7. contour_os   17. field
8. finish_as
9. finish_os

Symbol Definition Structure

```
                                          ┌──────────────────────┐
                                          │ BR_ROOT_BEVEL.SYM    │
                                          └──────────────────────┘
                                                    │
                                                    ▼
                    ┌──────────────┐       ┌──────────────────┐
                    │ ABOVE_REF    │◄──────│ ARROW_SIDE       │
                    │ BENEATH_REF  │       │ OTHER_SIDE       │
                    └──────────────┘       │ SYMMETRICAL      │
                            │              └──────────────────┘
  ┌──────────────┐         │                      │
  │ CONTINUOUS   │         ▼                       ▼
  │ INTERMITTENT │  ┌──────────────────┐  ┌──────────────────────┐
  └──────────────┘  │ WELD_SIZE        │  │ WELD_SIZE_AS         │
         ▲          │ STYLE            │  │ WELD_SIZE_OS         │
         │          │ CONTOUR          │  │ STYLE_AS             │
  ┌──────────────┐  │ BACK_TYPE        │  │ STYLE_OS             │
  │ FLAT         │  │ FINISH           │  │ CONTOUR_AS           │
  │ CONVEX       │  │ LEADER_ORIENT    │  │ CONTOUR_OS           │
  │ SMOOTH_BLEND │  └──────────────────┘  │ FINISH_AS            │
  └──────────────┘          ┌──────────────┐│ FINISH_OS           │
  ┌──────────────┐          │ CONTINUOUS   ││ LEADER_ORIENT       │
  │ CONTINUOUS   │          │ INTERMITTENT │└──────────────────────┘
  │ INTERMITTENT │          └──────────────┘           │
  └──────────────┘                                     ▼
         ▲               ┌──────────────┐       ┌──────────────┐
         │               │ BACK_SIZE    │       │ LEFT         │
         │               │ STYLE        │       │ RIGHT        │
         │               │ CONTOUR      │       └──────────────┘
         │               │ FINISH       │              │
         │               └──────────────┘              ▼
  ┌──────────────┐              │            ┌──────────────┐
  │ FLAT         │   ┌──────────────┐  ┌──────────┐ │ ALL_AROUND   │
  │ CONVEX       │   │ BACKING      │  │ LEFT     │ │ FIELD        │
  │ SMOOTH_BLEND │   │ STRIP        │  │ RIGHT    │ │ TAIL         │
  └──────────────┘   └──────────────┘  └──────────┘ └──────────────┘
                          │                  │             │
                          ▼                  ▼             ▼
                   ┌──────────────┐  ┌──────────────┐ ┌──────────────┐
                   │ PERMANENT    │  │ FLAT         │ │ REFERENCE    │
                   │ REMOVABLE    │  │ CONVEX       │ │ WELD_PROCESS │
                   └──────────────┘  │ SMOOTH_BLEND │ └──────────────┘
                                     └──────────────┘
                   ┌──────────────┐
                   │ ALL_AROUND   │
                   │ FIELD        │
                   │ TAIL         │
                   └──────────────┘
                          │
                          ▼
                   ┌──────────────┐
                   │ REFERENCE    │
                   │ WELD_PROCESS │
                   └──────────────┘
```

Slot Symbol: Iso_Slot.sym

EXAMPLE 1

PICKS

1. arrow_side
2. slot_width
3. style
4. contour
5. finish
6. leader-orient
7. intermittent
8. convex
9. left
10. field

11. tail
12. weld_process

6   10x40(100)   DFRQ45

EXAMPLE 2

PICKS

1. other_side
2. slot_width
3. style
4. contour
5. leader_orient
6. continuous
7. smooth_blend
8. left
9. field
10. tail

11. reference

5.5   400   HJKG45

Symbol Definition Structure

```
                                    ┌─────────────────┐
                                    │ ISO_SLOT.SYM    │
                                    └─────────────────┘
                                             │
                                             ▼
┌──────────────────┐              ┌─────────────────┐
│ CONTINUOUS       │◄─────┐       │ ARROW_SIDE      │
│ INTERMITTENT     │      │       │ OTHER_SIDE      │
└──────────────────┘      │       └─────────────────┘
                          │                │
┌──────────────────┐      │                ▼
│ FLAT             │      │       ┌─────────────────┐
│ CONVEX           │      │       │ SLOT_WIDTH      │
│ SMOOTH_BLEND     │◄─────┴───────│ STYLE           │
└──────────────────┘              │ CONTOUR         │
                                  │ FINISH          │
                                  │ LEADER_ORIENT   │
                          ┌───────└─────────────────┘
                          │
                          ▼
              ┌──────────────────┐
              │ LEFT             │
              │ RIGHT            │
              └──────────────────┘
                          │
                          ▼
              ┌──────────────────┐
              │ FIELD            │
              │ TAIL             │
              └──────────────────┘
                          │
                          ▼
              ┌──────────────────┐
              │ REFERENCE        │
              │ WELD_PROCESS     │
              └──────────────────┘
```
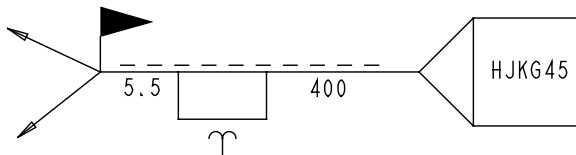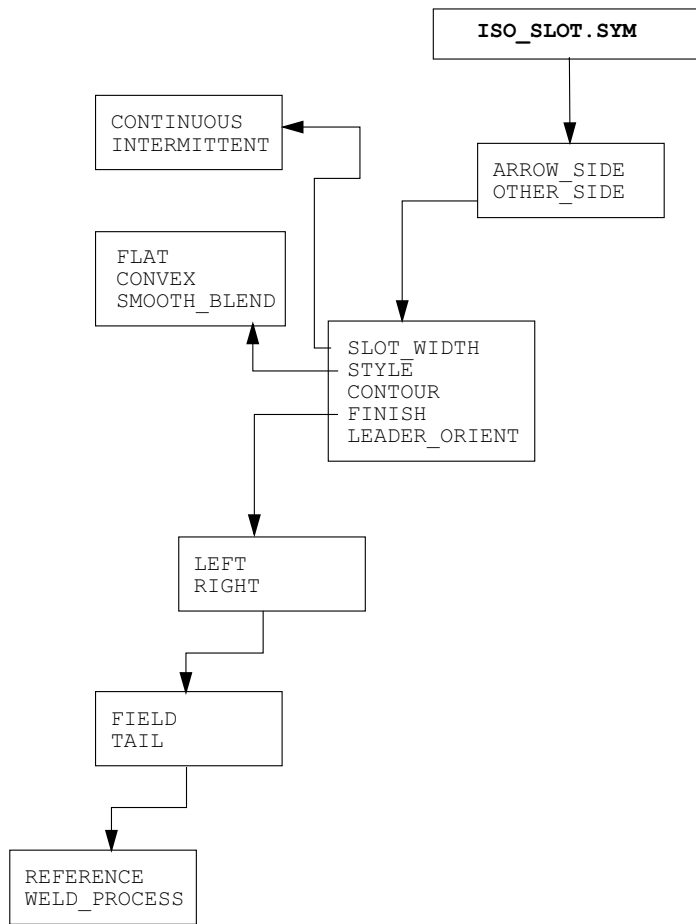
Spot Symbol: Iso_Spot.sym

EXAMPLE 1

```
PICKS

 1. fusion              11. tail
 2. above_ref           12. weld_process
 3. spot_dia
 4. number_pitch
 5. contour
 6. finish
 7. leader_orient
 8. convex
 9. left
10. field
```
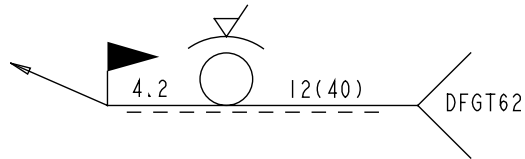

EXAMPLE 2

```
PICKS

 1. resistance
 2. spot_dia
 3. number_pitch
 4. leader_orient
 5. left
 6. field
 7. tail
 8. reference
```
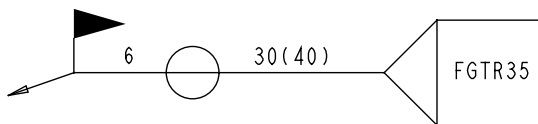
EXAMPLE 3

PICKS

```
1. projection
2. project_dia
3. number_pitch
4. leader_orient
5. left
6. tail
7. reference
```

P5.5   I2(40)   XDSC

Symbol Definition Structure

```
                                    ISO_SPOT.SYM


                    ABOVE_REF  <---------  FUSION
                    BENEATH_REF            RESISTANCE
                                           PROJECTION


        FLAT         SPOT_DIA         SPOT_DIA         PROJECT_DIA
        CONVEX       NUMBER_PITCH     NUMBER_PITCH     NUMBER_PITCH
        CONCAVE  <-- CONTOUR          LEADER_ORIENT    LEADER_ORIENT
        SMOOTH_BLEND FINISH
                     LEADER_ORIENT


                     LEFT             LEFT
                     RIGHT            RIGHT


                     FIELD            FIELD
                     TAIL             TAIL


                     REFERENCE        REFERENCE
                     WELD_PROCESS     WELD_PROCESS
```
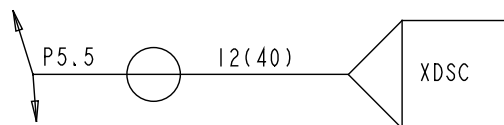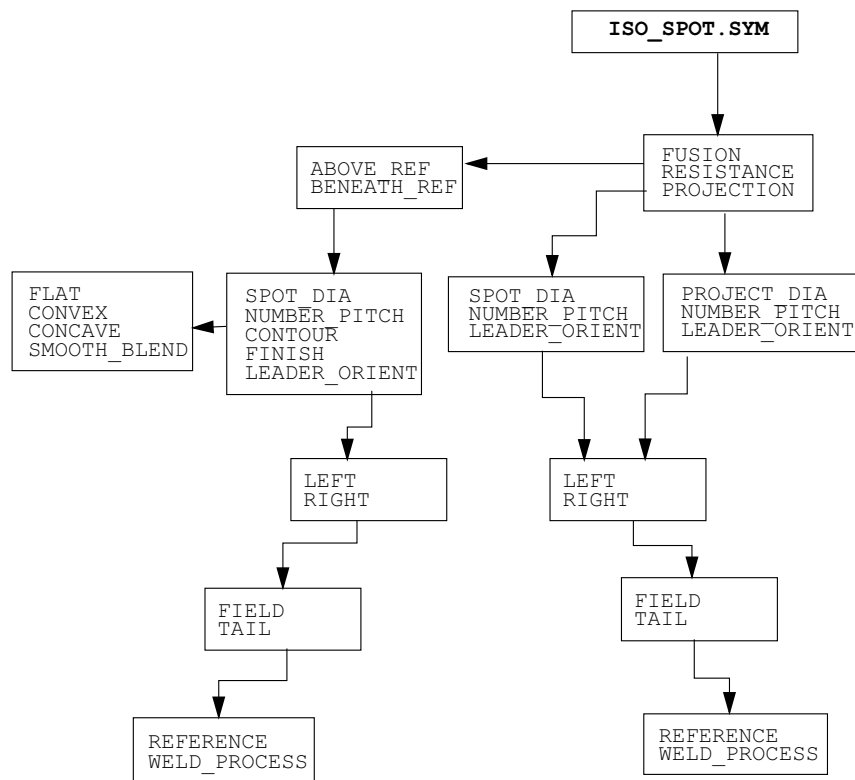
Seam Symbol: Iso_Seam.sym

EXAMPLE 1

PICKS

1. fusion	11. field
2. above_ref	12. tail
3. seam_width	13. weld_process
4. style	
5. contour	
6. finish	
7. leader_orient	
8. intermittent	
9. convex	
10. left	
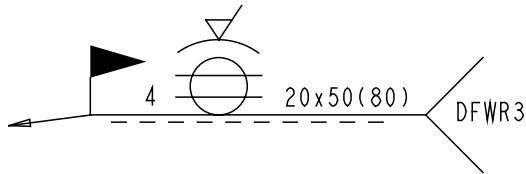
EXAMPLE 2

PICKS

1. resistance
2. seam_width
3. style
4. leader_orient
5. continuous
6. left
7. field
8. tail
9. reference
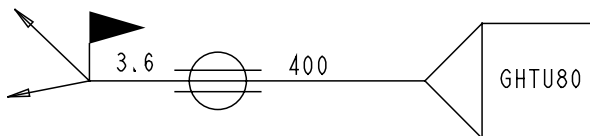
EXAMPLE 3

PICKS

1. projection
2. seam_width
3. style
4. intermittent

P5.5 ———⊖——— 20x50(80)

Symbol Definition Structure

ISO_SEAM.SYM

RESISTANCE
FUSION
PROJECTION

ABOVE_REF
BENEATH_REF

CONTINUOUS
INTERMITTENT

CONTOUR
FINISH
LEADER_ORIENT
SEAM_WIDTH
STYLE

FLAT
CONVEX
CONCAVE
SMOOTH_BLEND

SEAM_WIDTH
STYLE
LEADER_ORIENT

CONTINUOUS
INTERMITTENT

LEFT
RIGHT

LEFT
RIGHT

FIELD
TAIL

FIELD
TAIL

REFERENCE
WELD_PROCESS

REFERENCE
WELD_PROCESS
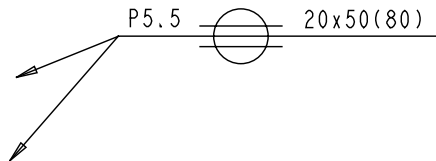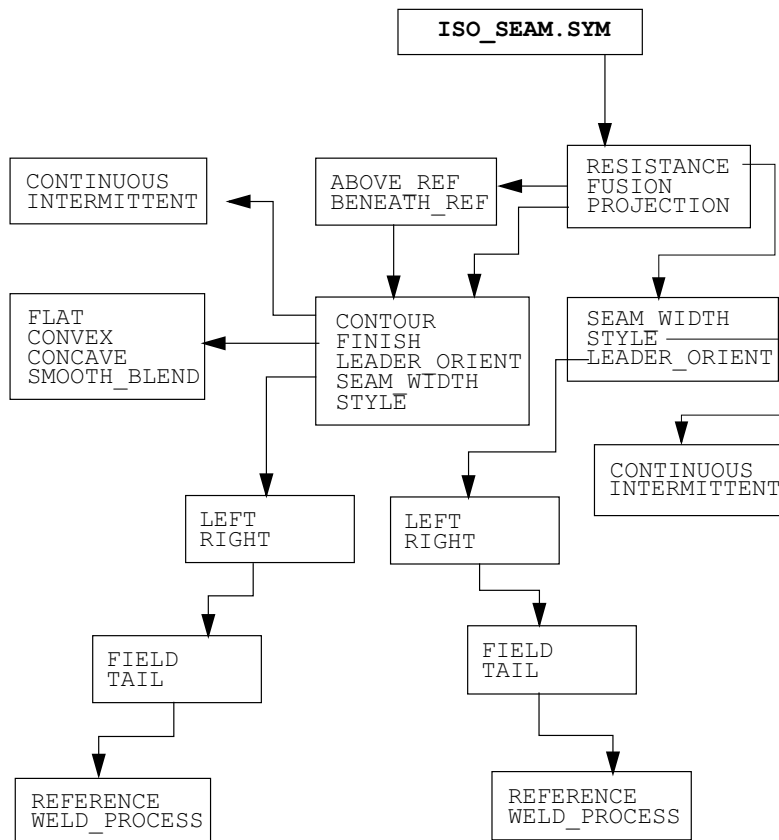
Plug Symbol: Iso_Plug.sym

EXAMPLE 1

PICKS

1. arrow_side   11. weld_process
2. hole_dia
3. number_space
4. contour
5. finish
6. leader_orient
7. convex
8. left
9. field
10. tail

5.5   I2 (30)   SDRT55
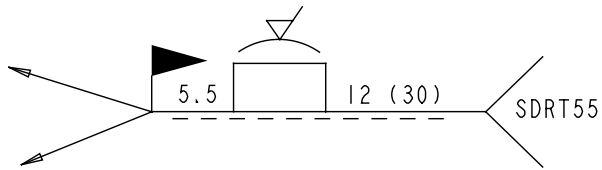
EXAMPLE 2

PICKS

1. other_side
2. hole_dia
3. contour
4. number_space
5. leader_orient
6. smooth_blend
7. left
8. field
9. tail
10. reference

3.2   I2 (30)   FGHJ45

Symbol Definition Structure

```
                          ┌─────────────────┐
                          │  ISO_PLUG.SYM   │
                          └─────────────────┘
                                   │
                                   ▼
                          ┌─────────────────┐
                          │ ARROW_SIDE      │
                          │ OTHER_SIDE      │
                          └─────────────────┘
                                 │
  ┌─────────────────┐            │
  │ FLAT            │            │
  │ CONVEX          │            ▼
  │ SMOOTH_BLEND    │   ┌─────────────────┐
  └─────────────────┘   │ HOLE_DIA        │
          ▲     ┌───────│ NUMBER_SPACE    │
          └─────┘       │ CONTOUR         │
                        │ FINISH          │
                ┌───────│ LEADER_ORIENT   │
                │       └─────────────────┘
                ▼
       ┌─────────────────┐
       │ LEFT            │
       │ RIGHT           │
       └─────────────────┘
              │
              ▼
       ┌─────────────────┐
       │ FIELD           │
       │ TAIL            │
       └─────────────────┘
           │
           ▼
   ┌─────────────────┐
   │ REFERENCE       │
   │ WELD_PROCESS    │
   └─────────────────┘
```
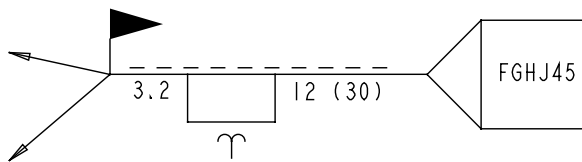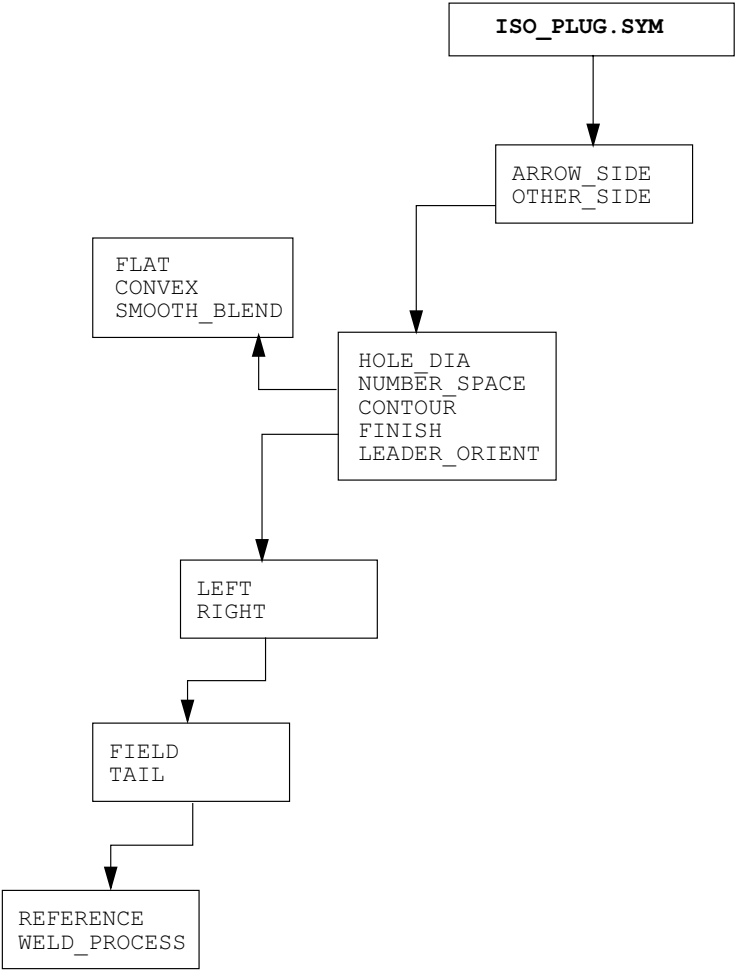
Fillet Symbol: Iso_Fillet.sym

EXAMPLE 1

```
PICKS

1. arrow_side        8. intermittent
2. above_ref         9. convex
3. fillet_size      10. left
4. style            11. field
5. contour          12. all_around
6. finish           13. tail
7. leader_orient    14. weld_process
```



```
                    4    10x50(100)    DFXS32
```
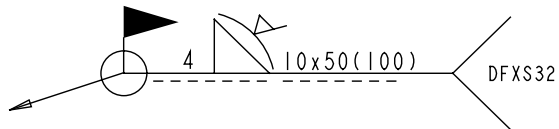
EXAMPLE 2

```
PICKS

1. other_side        8. flat
2. beneath_ref       9. left
3. fillet_size      10. field
4. style            11. all_around
5. contour          12. tail
6. leader_orient    13. reference
7. continuous
```



```
        6        400        DWRT45
```
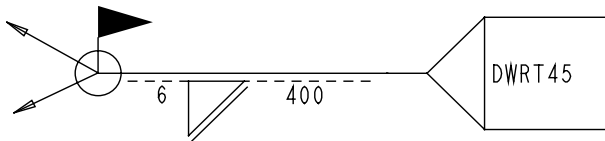
EXAMPLE 3

PICKS

1. symmetrical   11. intermittent
2. fil_size_as   12. continuous
3. fil_size_os   13. concave
4. style_as   14. flat
5. style_os   15. left
6. contour_as   16. all_around
7. contour_os   17. field
8. finish_as
9. finish_os
10. leader_orient
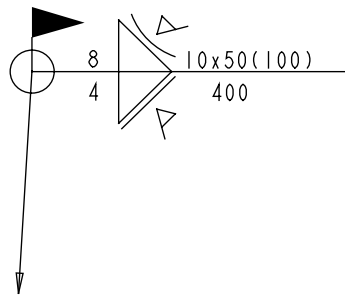
EXAMPLE 4

PICKS

1. staggered
2. fil_size_as
3. contour_as
4. leader_orient
5. smooth_blend
6. left
7. all_around
8. field

Symbol Definition Structure

```
                                    ┌─────────────────┐
                                    │  ISO_FILLET.SYM │
                                    └─────────────────┘
                                              │
                                              ▼
┌──────────────┐      ┌──────────────┐      ┌──────────────┐
│ CONTINUOUS   │◄─────│ ABOVE_REF    │◄─────│ ARROW_SIDE   │
│ INTERMITTENT │      │ BENEATH_REF  │      │ OTHER_SIDE   │
└──────────────┘      └──────────────┘      │ SYMMETRICAL  │
                                            │ STAGGERED    │
                                            └──────────────┘
                              │                     │
                              ▼                     ▼
┌──────────────┐      ┌──────────────┐      ┌──────────────┐
│ FLAT         │◄─────│ FILLET_SIZE  │      │ FIL_SIZE_AS  │
│ CONVEX       │      │ STYLE        │      │ FIL_SIZE_OS  │
│ CONCAVE      │      │ CONTOUR      │      │ STYLE_OS     │
│ SMOOTH_BLEND │      │ FINISH       │      │ CONTOUR_AS   │
└──────────────┘      │ LEADER_ORIENT│      │ CONTOUR_OS   │
                      └──────────────┘      │ FINISH_AS    │
                         │                  │ FINISH_OS    │
                         │                  │ LEADER_ORIENT│
                         ▼                  └──────────────┘
              ┌──────────────┐                    │
              │ LEFT         │      ┌──────────────┐
              │ RIGHT        │      │ CONTINUOUS   │   ┌──────────────┐
              └──────────────┘      │ INTERMITTENT │   │ LEFT         │
                   │                └──────────────┘   │ RIGHT        │
                   ▼                      │            └──────────────┘
        ┌──────────────┐         ┌──────────────┐          │
        │ FIELD        │         │ FLAT         │          ▼
        │ ALL_AROUND   │         │ CONVEX       │   ┌──────────────┐
        │ TAIL         │         │ CONCAVE      │   │ FIELD        │
        └──────────────┘         │ SMOOTH_BLEND │   │ ALL_AROUND   │
             │                   └──────────────┘   │ TAIL         │
             ▼                                      └──────────────┘
    ┌──────────────┐                                     │
    │ REFERENCE    │                                     ▼
    │ WELD_PROCESS │                            ┌──────────────┐
    └──────────────┘                            │ REFERENCE    │
                                                │ WELD_PROCESS │
                                                └──────────────┘
```

J Butt Symbol: J_Butt.sym

EXAMPLE 1

PICKS

1. arrow_side	11. strip
2. above_ref	12. left
3. weld_size	13. removable
4. style	14. field
5. contour	15. all_around
6. back_type	16. tail
7. finish	17. weld_process
8. leader_orient	
9. intermittent	
10. convex	

EXAMPLE 2

PICKS

1. other_side	11. style
2. beneath_ref	12. contour
3. style	13. finish
4. contour	14. intermittent
5. back_type	15. smooth_blend
6. leader_orient	16. left
7. continuous	17. field
8. flat	18. all_around
9. backing	19. tail
10. back_size	20. reference

EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient

11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field

Symbol Definition Structure

J_BUTT.SYM

ABOVE_REF
BENEATH_REF

ARROW_SIDE
OTHER_SIDE
SYMMETRICAL

CONTINUOUS
INTERMITTENT

WELD_SIZE
STYLE
CONTOUR
BACK_TYPE
FINISH
LEADER_ORIENT

CONTINUOUS
INTERMITTENT

FLAT
CONVEX
SMOOTH_BLEND

WELD_SIZE_AS
WELD_SIZE_OS
STYLE_AS
STYLE_OS
CONTOUR_AS
CONTOUR_OS
FINISH_AS
FINISH_OS
LEADER_ORIENT

CONTINUOUS
INTERMITTENT

BACK_SIZE
STYLE
CONTOUR
FINISH

FLAT
CONVEX
SMOOTH_BLEND

BACKING
STRIP

LEFT
RIGHT

LEFT
RIGHT

PERMANENT
REMOVABLE

FLAT
CONVEX
SMOOTH_BLEND

ALL_AROUND
FIELD
TAIL

ALL_AROUND
FIELD
TAIL

REFERENCE
WELD_PROCESS

REFERENCE
WELD_PROCESS

Square Butt Symbol: Iso_Square.sym

EXAMPLE 1

PICKS

```
 1. arrow_side        11. strip
 2. above_ref         12. left
 3. weld_size         13. removable
 4. style             14. field
 5. contour           15. all_around
 6. back_type         16. tail
 7. finish            17. weld_process
 8. leader_orient
 9. intermittent
10. convex
```

```
        ▽
        ⌣
   ▶  ┃┃
 ○──┼┼┼┼──────────╲
 4  20x40(100)      ╲ SPG25
  ┌──┐
  │MR│
  └──┘
```

EXAMPLE 2

PICKS

```
 1. other_side       11. style
 2. beneath_ref      12. contour
 3. style            13. finish
 4. contour          14. intermittent
 5. back_type        15. smooth_blend
 6. leader_orient    16. left
 7. continuous       17. field
 8. flat             18. all_around
 9. backing          19. tail
10. back_size        20. reference
```

```
   ▶    ⌣
 ○──┼───⌒──────┌──────┐
 4  10x50(100) │SPG46 │
       200     └──────┘
      ┃┃
       △
```

EXAMPLE 3

PICKS

1. symmetrical	11. intermittent
2. weld_size_as	12. continuous
3. weld_size_os	13. convex
4. style_as	14. flat
5. style_os	15. left
6. contour_as	16. all_around
7. contour_os	17. field
8. finish_as	
9. finish_os	
10. leader_orient	

6   10x50(100)

4   200

Symbol Definition Structure

```
                                          ┌─────────────────┐
                                          │ ISO_SQUARE.SYM  │
                                          └────────┬────────┘
                                                   │
                                                   ▼
                    ┌──────────────┐      ┌─────────────────┐
                    │ ABOVE_REF    │◄─────│ ARROW_SIDE      │
                    │ BENEATH_REF  │      │ OTHER_SIDE      │
                    └──────┬───────┘      │ SYMMETRICAL     │
                           │              └─────────────────┘
    ┌──────────────┐       ▼                       │
    │ CONTINUOUS   │  ┌──────────────┐             │
    │ INTERMITTENT │  │ WELD_SIZE    │             ▼
    └──────────────┘  │ STYLE        │   ┌──────────────────┐
              ▲       │ CONTOUR      │   │ WELD_SIZE_AS     │
              │       │ BACK_TYPE    │   │ WELD_SIZE_OS     │
      ┌──────────────┐│ FINISH       │   │ STYLE_AS         │
      │ FLAT         ││ LEADER_ORIENT│   │ STYLE_OS         │
      │ CONVEX       │└──────────────┘   │ CONTOUR_AS       │
      │ SMOOTH_BLEND │                   │ CONTOUR_OS       │
      └──────────────┘ ┌──────────────┐ │ FINISH_AS        │
  ┌──────────────┐     │ CONTINUOUS   │◄│ FINISH_OS        │
  │ CONTINUOUS   │     │ INTERMITTENT │ │ LEADER_ORIENT    │
  │ INTERMITTENT │     └──────────────┘ └──────────────────┘
  └──────────────┘                               │
        ▲      ┌──────────────┐                  ▼
        │      │ BACK_SIZE    │        ┌──────────────┐
        │      │ STYLE        │        │ LEFT         │
        │      │ CONTOUR      │        │ RIGHT        │
        │      │ FINISH       │        └──────────────┘
        │      └──────────────┘                  │
  ┌──────────────┐   ┌──────────┐  ┌────────┐    ▼
  │ FLAT         │   │ BACKING  │  │ LEFT   │ ┌──────────────┐
  │ CONVEX       │   │ STRIP    │  │ RIGHT  │ │ ALL_AROUND   │
  │ SMOOTH_BLEND │   └──────────┘  └────────┘ │ FIELD        │
  └──────────────┘        ▲                   │ TAIL         │
                ┌──────────────┐              └──────────────┘
                │ PERMANENT    │  ┌──────────────┐     │
                │ REMOVABLE    │  │ FLAT         │     ▼
                └──────────────┘  │ CONVEX       │ ┌──────────────┐
                                  │ SMOOTH_BLEND │ │ REFERENCE    │
              ┌──────────────┐    └──────────────┘ │ WELD_PROCESS │
              │ ALL_AROUND   │                     └──────────────┘
              │ FIELD        │
              │ TAIL         │
              └──────────────┘
                   │
                   ▼
           ┌──────────────┐
           │ REFERENCE    │
           │ WELD_PROCESS │
           └──────────────┘
```

V Butt Symbol: V_Butt.sym

EXAMPLE 1

PICKS

```
 1. arrow_side          11. left
 2. above_ref           12. removable
 3. style               13. field
 4. contour             14. all_around
 5. back_type           15. tail
 6. finish              16. weld_process
 7. leader_orient
 8. intermittent
 9. convex
10. strip
```

10X50(100)   DFRW

MR

EXAMPLE 2

PICKS

```
 1. other_side          11. style
 2. beneath_ref         12. contour
 3. style               13. finish
 4. contour             14. intermittent
 5. back_type           15. smooth_blend
 6. leader_orient       16. left
 7. continuous          17. field
 8. flat                18. all_around
 9. backing             19. tail
10. back_size           20. reference
```

3   6x30(80)   GHTR
350

EXAMPLE 3

PICKS

1. symmetrical	11. convex
2. style_as	12. flat
3. style_os	13. left
4. contour_as	14. all_around
5. contour_os	15. field
6. finish_as	
7. finish_os	
8. leader_orient	
9. intermittent	
10. continuous	

6 x 60 (120)
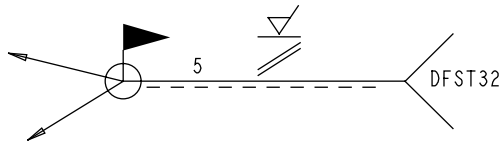
500

Symbol Definition Structure

```
                                                    ┌─────────────┐
                                                    │ V_BUTT.SYM  │
                                                    └──────┬──────┘
                                                           │
                                                           ▼
                              ┌──────────────┐      ┌──────────────┐
                              │ ABOVE_REF    │◄─────│ ARROW_SIDE   │
                              │ BENEATH_REF  │      │ OTHER_SIDE   │
                              └──────┬───────┘      │ SYMMETRICAL  │
                                     │              └──────────────┘
                                     ▼
        ┌──────────────┐     ┌──────────────┐
        │ CONTINUOUS   │◄────│ STYLE        │
        │ INTERMITTENT │     │ CONTOUR      │
        └──────────────┘     │ BACK_TYPE    │
                             │ FINISH       │
                             │ LEADER_ORIENT│
              ┌──────────────┐└──────────────┘
              │ FLAT         │              ┌──────────────┐    ┌──────────────┐
              │ CONVEX       │              │ CONTINUOUS   │◄───│ FINISH_AS    │
┌──────────────┐SMOOTH_BLEND │              │ INTERMITTENT │    │ FINISH_OS    │
│ CONTINUOUS   │└──────────────┘            └──────────────┘    │ STYLE_AS     │
│ INTERMITTENT │                                                │ STYLE_OS     │
└──────────────┘                                                │ CONTOUR_AS   │
       ▲                                                        │ CONTOUR_OS   │
       │          ┌──────────────┐                              │ LEADER_ORIENT│
       │          │ BACK_SIZE    │                              └──────┬───────┘
       │          │ STYLE        │                                     │
       │          │ CONTOUR      │                                     ▼
       │          │ FINISH       │                              ┌──────────────┐
       │          └──────────────┘                              │ LEFT         │
       │                                                        │ RIGHT        │
┌──────────────┐  ┌──────────────┐  ┌──────────────┐            └──────┬───────┘
│ FLAT         │  │ BACKING      │  │ LEFT         │                   │
│ CONVEX       │  │ STRIP        │  │ RIGHT        │                   ▼
│ SMOOTH_BLEND │  └──────────────┘  └──────────────┘            ┌──────────────┐
└──────────────┘         ▲                                      │ ALL_AROUND   │
                         │          ┌──────────────┐            │ FIELD        │
              ┌──────────────┐      │ FLAT         │            │ TAIL         │
              │ PERMANENT    │      │ CONVEX       │            └──────┬───────┘
              │ REMOVABLE    │      │ SMOOTH_BLEND │                   │
              └──────────────┘      └──────────────┘                   ▼
                                                               ┌──────────────┐
                    ┌──────────────┐                           │ REFERENCE    │
                    │ ALL_AROUND   │                           │ WELD_PROCESS │
                    │ FIELD        │                           └──────────────┘
                    │ TAIL         │
                    └──────────────┘
         ┌──────────────┐
         │ REFERENCE    │
         │ WELD_PROCESS │
         └──────────────┘
```

U Butt Symbol: U_Butt.sym

EXAMPLE 1

PICKS

```
 1. arrow_side        11. strip
 2. above_ref         12. left
 3. weld_size         13. removable
 4. style             14. field
 5. contour           15. all_around
 6. back_type         16. tail
 7. finish            17. weld_process
 8. leader_orient
 9. intermittent
10. convex
```
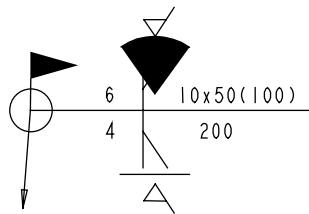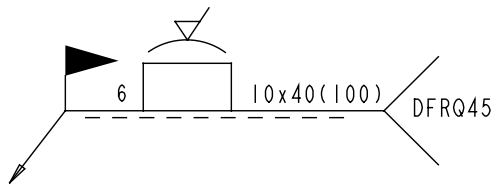

EXAMPLE 2

PICKS

```
 1. other_side        11. style
 2. beneath_ref       12. contour
 3. style             13. finish
 4. contour           14. intermittent
 5. back_type         15. smooth_blend
 6. leader_orient     16. left
 7. continuous        17. field
 8. flat              18. all_around
 9. backing           19. tail
10. back_size         20. reference
```

EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient

11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field

8   20x30(60)

6   400

Symbol Definition Structure

```
                                              ┌──────────────┐
                                              │  U_BUTT.SYM  │
                                              └──────┬───────┘
                                                     │
                          ┌─────────────┐     ┌──────▼───────┐
                          │ ABOVE_REF   │◄────│ ARROW_SIDE   │
                          │ BENEATH_REF │     │ OTHER_SIDE   │
                          └──────┬──────┘     │ SYMMETRICAL  │
                                 │            └──────┬───────┘
      ┌──────────────┐           │                   │
      │ CONTINUOUS   │           ▼                   │
      │ INTERMITTENT │    ┌──────────────┐           │
      └──────────────┘    │ WELD_SIZE    │           │
                          │ STYLE        │           │
      ┌──────────────┐    │ CONTOUR      │           │
      │ FLAT         │    │ BACK_TYPE    │    ┌───────▼────────┐
      │ CONVEX       │    │ FINISH       │    │ WELD_SIZE_AS   │
      │ SMOOTH_BLEND │    │ LEADER_ORIENT│    │ WELD_SIZE_OS   │
      └──────────────┘    └──────────────┘    │ STYLE_AS       │
  ┌──────────────┐                            │ STYLE_OS       │
  │ CONTINUOUS   │        ┌──────────────┐    │ CONTOUR_AS     │
  │ INTERMITTENT │        │ CONTINUOUS   │◄───│ CONTOUR_OS     │
  └──────────────┘        │ INTERMITTENT │    │ FINISH_AS      │
                          └──────────────┘    │ FINISH_OS      │
      ┌──────────────┐                        │ LEADER_ORIENT  │
      │ BACK_SIZE    │                        └───────┬────────┘
      │ STYLE        │                                │
      │ CONTOUR      │                        ┌───────▼────────┐
      │ FINISH       │                        │ LEFT           │
      └──────────────┘                        │ RIGHT          │
                                              └───────┬────────┘
  ┌──────────────┐  ┌──────────────┐  ┌──────┐        │
  │ FLAT         │  │ BACKING      │  │ LEFT │ ┌───────▼────────┐
  │ CONVEX       │  │ STRIP        │  │ RIGHT│ │ ALL_AROUND     │
  │ SMOOTH_BLEND │  └──────────────┘  └──────┘ │ FIELD          │
  └──────────────┘                             │ TAIL           │
                                               └───────┬────────┘
            ┌──────────────┐  ┌──────────────┐         │
            │ PERMANENT    │  │ FLAT         │ ┌───────▼────────┐
            │ REMOVABLE    │  │ CONVEX       │ │ REFERENCE      │
            └──────────────┘  │ SMOOTH_BLEND │ │ WELD_PROCESS   │
                              └──────────────┘ └────────────────┘
              ┌──────────────┐
              │ ALL_AROUND   │
              │ FIELD        │
              │ TAIL         │
  ┌──────────────┐
  │ REFERENCE    │
  │ WELD_PROCESS │
  └──────────────┘
```

Surface Joint Symbol: Surface_Joint.sym

EXAMPLE 1

PICKS

```
1. arrow_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. weld_process
```


EXAMPLE 2

PICKS

```
1. other_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference
```

Symbol Definition Structure

```
        ┌─────────────────────────┐
        │  SURFACE_JOINT.SYM       │
        └─────────────────────────┘
                      │
                      ▼
        ┌─────────────────────────┐
        │  ARROW_SIDE              │
        │  OTHER_SIDE              │
        └─────────────────────────┘
              │
              ▼
        ┌─────────────────────────┐
        │  WELD_SIZE               │
        │  LEADER_ORIENT           │
        └─────────────────────────┘
              │
              ▼
        ┌─────────────────────────┐
        │  LEFT                    │
        │  RIGHT                   │
        └─────────────────────────┘
              │
              ▼
        ┌─────────────────────────┐
        │  ALL_AROUND              │
        │  FIELD                   │
        │  TAIL                    │
        └─────────────────────────┘
              │
              ▼
        ┌─────────────────────────┐
        │  REFERENCE               │
        │  WELD_PROCESS            │
        └─────────────────────────┘
```
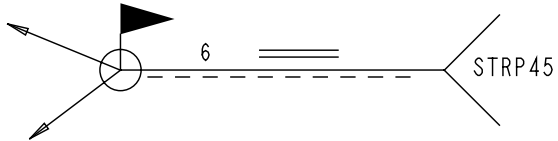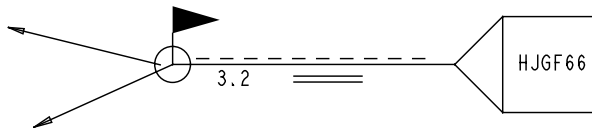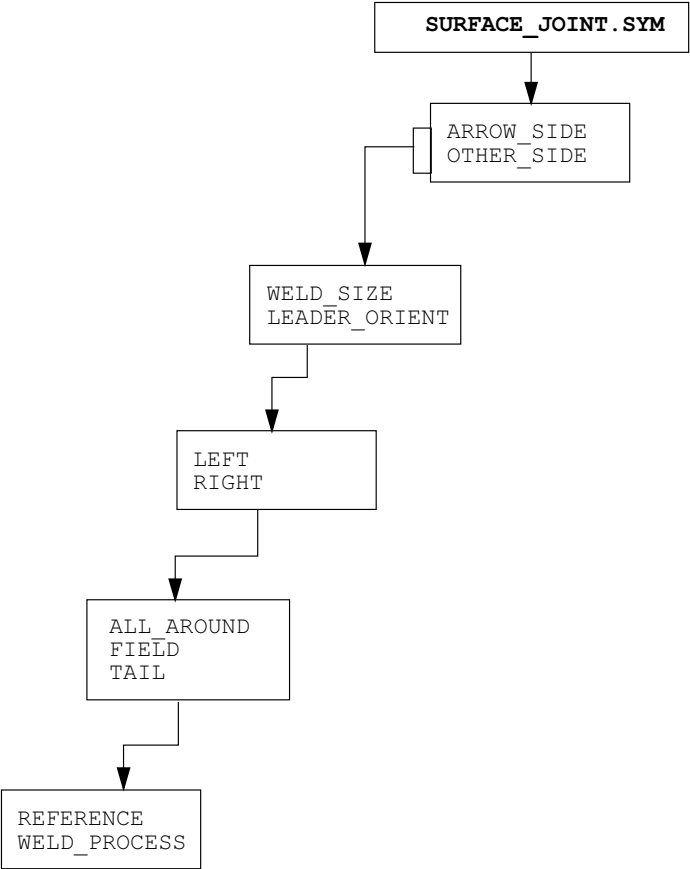
Surfacing Symbol: Iso_Surfacing.sym

EXAMPLE 1

PICKS

```
 1. arrow_side        11. weld_process
 2. weld_size
 3. contour
 4. finish
 5. leader_orient
 6. concave
 7. left
 8. field
 9. all_around
10. tail
```
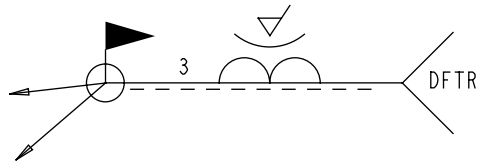
EXAMPLE 2

PICKS

```
 1. other_side
 2. weld_size
 3. contour
 4. leader_orient
 5. flat
 6. left
 7. field
 8. all_around
 9. tail
10. reference
```

Symbol Definition Structure

```
                              ┌─────────────────────┐
                              │  ISO_SURFACING.SYM  │
                              └─────────────────────┘
                                        │
                                        ▼
                              ┌─────────────────────┐
                            ┌─│  ARROW_SIDE         │
                            │ │  OTHER_SIDE         │
                            │ └─────────────────────┘
  ┌─────────────────────┐   │
  │  FLAT               │   │
  │  CONVEX             │   │
  │  CONCAVE            │   ▼
  │  SMOOTH_BLEND       │ ┌─────────────────────┐
  └─────────────────────┘ │  WELD_SIZE          │
            ▲           ┌─│  CONTOUR            │
            │           │ │  FINISH             │
            └───────────┘ │  LEADER_ORIENT      │
                      ┌───└─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │  LEFT               │
            │  RIGHT              │
            └─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │  ALL_AROUND         │
            │  FIELD              │
            │  TAIL               │
            └─────────────────────┘
                      │
                      ▼
            ┌─────────────────────┐
            │  REFERENCE          │
            │  WELD_PROCESS       │
            └─────────────────────┘
```
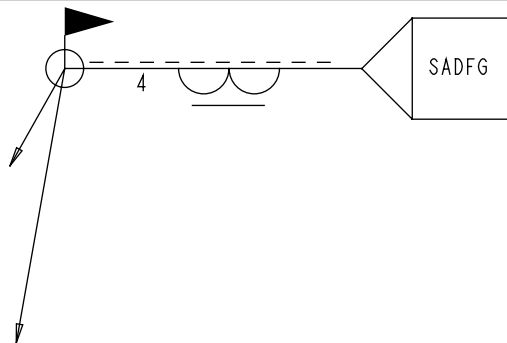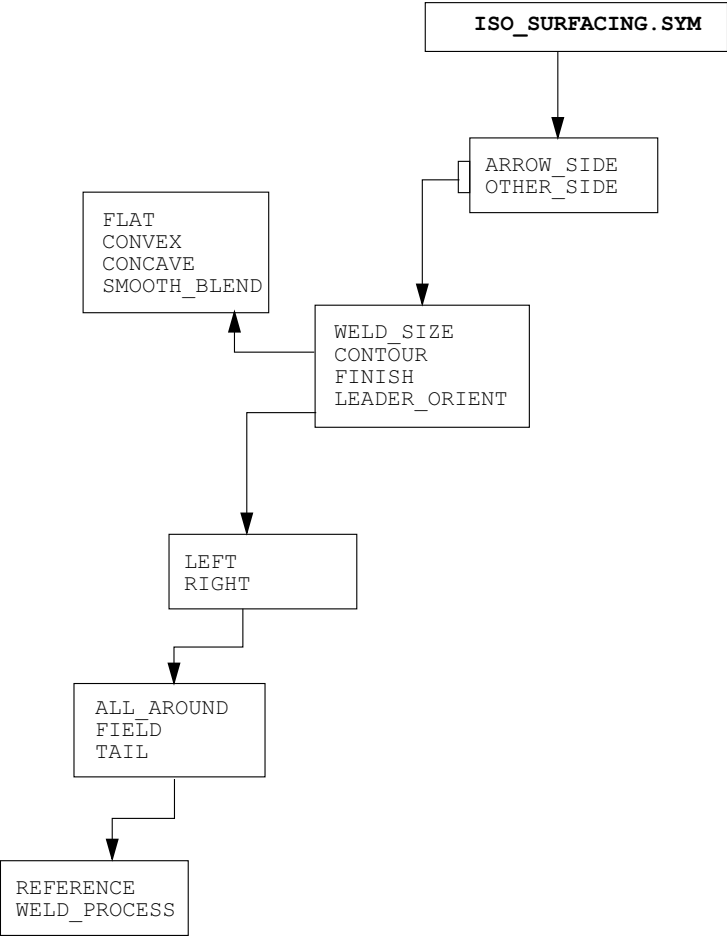
Steep-flanked V-butt Symbol: Steep_V.sym

EXAMPLE 1

PICKS

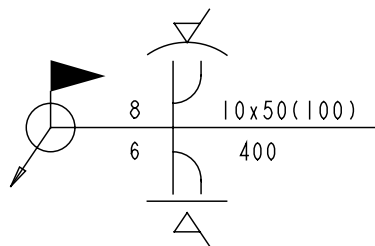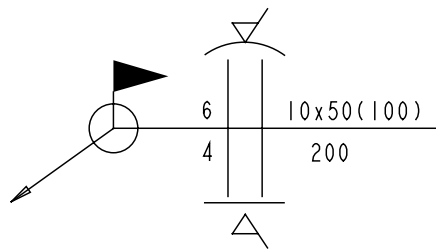|	
1. arrow_side	11. left
2. above_ref	12. removable
3. style	13. field
4. contour	14. all_around
5. back_type	15. tail
6. finish	16. weld_process
7. leader_orient	
8. intermittent	
9. convex	
10. strip	

10x50(100)   ASTM42

MR

EXAMPLE 2

PICKS

1. other_side	11. style
2. beneath_ref	12. contour
3. style	13. finish
4. contour	14. intermittent
5. back_type	15. smooth_blend
6. leader_orient	16. left
7. continuous	17. field
8. flat	18. all_around
9. backing	19. tail
10. back_size	20. reference

3   10x50(100)   DFRT30
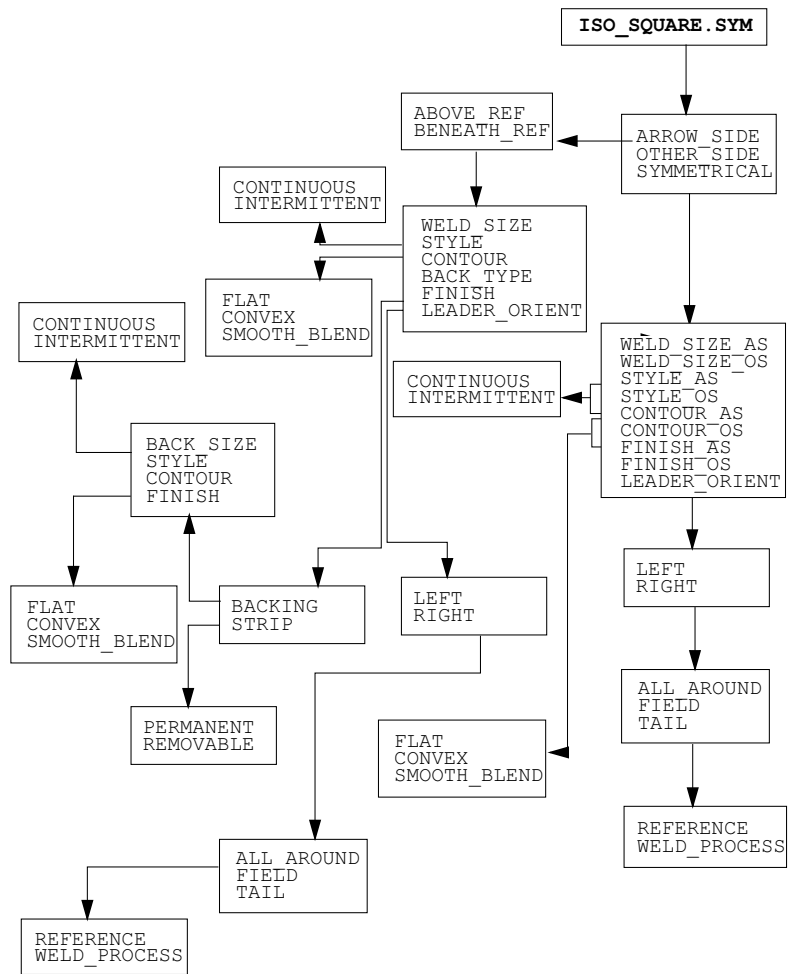
400

EXAMPLE 3

PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent
10. continuous

11. convex
12. flat
13. left
14. all_around
15. field

10x50(100)
400

Symbol Definition Structure

```
                                          ┌─────────────────┐
                                          │  STEEP_V.SYM    │
                                          └─────────────────┘
                                                  │
                                                  ▼
                    ┌──────────────┐      ┌─────────────────┐
                    │ ABOVE_REF    │◄─────│  ARROW_SIDE     │
                    │ BENEATH_REF  │      │  OTHER_SIDE     │
                    └──────────────┘      │  SYMMETRICAL    │
                          │               └─────────────────┘
                          ▼                       │
   ┌──────────────┐  ┌──────────────┐             │
   │ CONTINUOUS   │◄─│ STYLE        │             │
   │ INTERMITTENT │  │ CONTOUR      │             │
   └──────────────┘  │ FINISH       │             ▼
                     │ BACK_TYPE    │      ┌─────────────────┐
          ┌──────────│ LEADER_ORIENT│      │  FINISH_AS      │
          ▼          └──────────────┘      │  FINISH_OS      │
   ┌──────────────┐                        │  STYLE_AS       │
   │ FLAT         │                        │  STYLE_OS       │
   │ CONVEX       │   ┌──────────────┐     │  CONTOUR_AS     │
   │ SMOOTH_BLEND │   │ CONTINUOUS   │◄────│  CONTOUR_OS     │
   └──────────────┘   │ INTERMITTENT │     │  LEADER_ORIENT  │
┌──────────────┐      └──────────────┘     └─────────────────┘
│ CONTINUOUS   │                                   │
│ INTERMITTENT │                                   ▼
└──────────────┘      ┌──────────────┐      ┌─────────────────┐
      ▲               │ BACK_SIZE    │      │ LEFT            │
      │               │ STYLE        │      │ RIGHT           │
      │               │ CONTOUR      │      └─────────────────┘
      │               │ FINISH       │             │
      │               └──────────────┘             ▼
┌──────────────┐  ┌──────────────┐ ┌──────────┐  ┌─────────────────┐
│ FLAT         │  │ BACKING      │ │ LEFT     │  │ ALL_AROUND      │
│ CONVEX       │  │ STRIP        │ │ RIGHT    │  │ FIELD           │
│ SMOOTH_BLEND │  └──────────────┘ └──────────┘  │ TAIL            │
└──────────────┘                                 └─────────────────┘
         ┌──────────────┐                               │
         │ PERMANENT    │   ┌──────────────┐            ▼
         │ REMOVABLE    │   │ FLAT         │     ┌─────────────────┐
         └──────────────┘   │ CONVEX       │     │ REFERENCE       │
                            │ SMOOTH_BLEND │     │ WELD_PROCESS    │
            ┌──────────────┐└──────────────┘     └─────────────────┘
            │ ALL_AROUND   │
            │ FIELD        │
            │ TAIL         │
            └──────────────┘
   ┌──────────────┐
   │ REFERENCE    │
   │ WELD_PROCESS │
   └──────────────┘
```

Steep-flanked Bevel Butt Symbol: Steep_Bevel.sym

EXAMPLE 1

PICKS
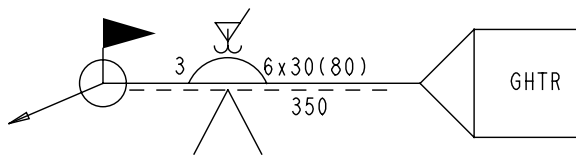
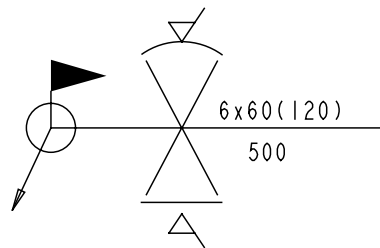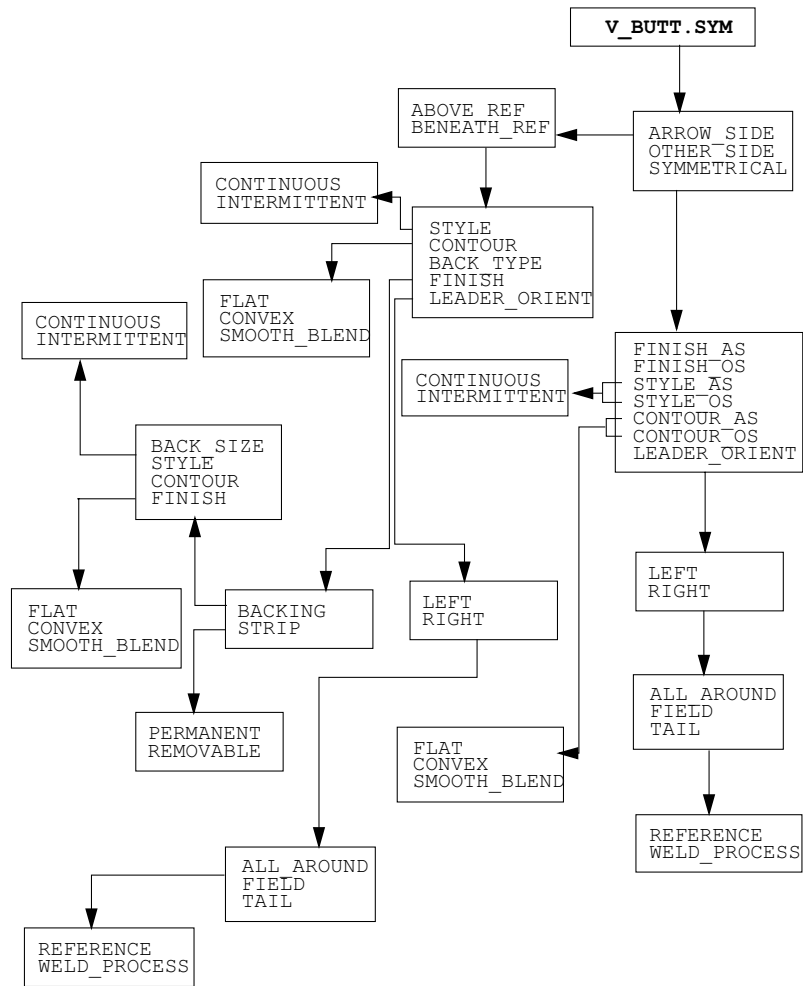|	
1. arrow_side	11. left
2. above_ref	12. removable
3. style	13. field
4. contour	14. all_around
5. back_type	15. tail
6. finish	16. weld_process
7. leader_orient	
8. intermittent	
9. convex	
10. strip	

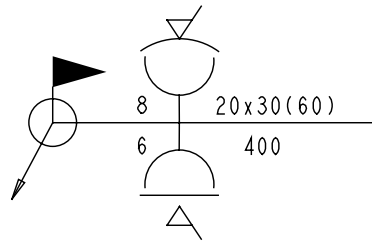

EXAMPLE 2

PICKS

1. other_side	11. style
2. beneath_ref	12. contour
3. style	13. finish
4. contour	14. intermittent
5. back_type	15. smooth_blend
6. leader_orient	16. left
7. continuous	17. field
8. flat	18. all_around
9. backing	19. tail
10. back_size	20. reference

EXAMPLE 3

PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent

11. convex
12. flat
13. left
14. all_around
15. field

l0x50(l00)
500

Symbol Definition Structure

```
                                          ┌─────────────────┐
                                          │ STEEP_BEVEL.SYM │
                                          └─────────────────┘
                                                   │
                                                   ▼
              ┌─────────────┐              ┌─────────────┐
              │ ABOVE_REF   │◄─────────────│ ARROW_SIDE  │
              │ BENEATH_REF │              │ OTHER_SIDE  │
              └─────────────┘              │ SYMMETRICAL │
                     │                     └─────────────┘
                     ▼                            │
  ┌─────────────┐ ┌───────────────┐               │
  │ CONTINUOUS  │◄│ STYLE         │               │
  │ INTERMITTENT│ │ CONTOUR       │               ▼
  └─────────────┘ │ BACK_TYPE     │        ┌──────────────┐
                  │ FINISH        │        │ FINISH_AS    │
       ┌──────────│ LEADER_ORIENT │        │ FINISH_OS    │
       │          └───────────────┘        │ STYLE_AS     │
       ▼                                   │ STYLE_OS     │
  ┌──────────────┐              ┌──────────┤ CONTOUR_AS   │
  │ FLAT         │       ┌──────┤CONTINUOUS│◄│ CONTOUR_OS  │
  │ CONVEX       │       │      │INTERMIT. │ │ LEADER_ORIENT│
  │ SMOOTH_BLEND │       │      └──────────┘ └──────────────┘
  └──────────────┘       │           │              │
                         │           │              ▼
  ┌──────────────┐       │           │       ┌──────────┐
  │ CONTINUOUS   │       │           │       │ LEFT     │
  │ INTERMITTENT │       │           │       │ RIGHT    │
  └──────────────┘       │           │       └──────────┘
         ▲               │           │              │
         │               ▼           ▼              ▼
  ┌──────────────┐ ┌──────────┐ ┌──────────┐ ┌──────────────┐
  │ BACK_SIZE    │ │ BACKING  │ │ LEFT     │ │ ALL_AROUND   │
  │ STYLE        │ │ STRIP    │ │ RIGHT    │ │ FIELD        │
  │ CONTOUR      │ └──────────┘ └──────────┘ │ TAIL         │
  │ FINISH       │                           └──────────────┘
  └──────────────┘                                  │
         ▲                                          ▼
         │                              ┌──────────────┐ ┌──────────────┐
  ┌──────────────┐ ┌──────────────┐     │ FLAT         │ │ REFERENCE    │
  │ FLAT         │ │ PERMANENT    │     │ CONVEX       │ │ WELD_PROCESS │
  │ CONVEX       │ │ REMOVABLE    │     │ SMOOTH_BLEND │ └──────────────┘
  │ SMOOTH_BLEND │ └──────────────┘     └──────────────┘
  └──────────────┘
                    ┌──────────────┐
                    │ ALL_AROUND   │
                    │ FIELD        │
                    │ TAIL         │
                    └──────────────┘
                           │
                           ▼
                    ┌──────────────┐
                    │ REFERENCE    │
                    │ WELD_PROCESS │
                    └──────────────┘
```
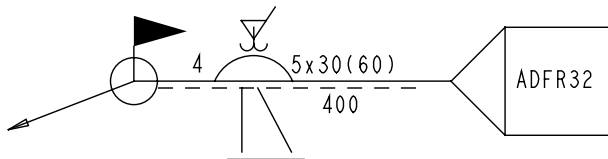
V-butt with Broad Root Face Symbol: Broad_Root_V.sym

EXAMPLE 1

PICKS

```
 1. arrow_side        11. strip
 2. above_ref         12. left
 3. weld_size         13. removable
 4. style             14. field
 5. contour           15. all_around
 6. back_type         16. tail
 7. finish            17. weld_process
 8. leader_orient
 9. intermittent
10. convex
```


EXAMPLE 2

PICKS

```
 1. other_side        11. style
 2. beneath_ref       12. contour
 3. style             13. finish
 4. contour           14. intermittent
 5. back_type         15. smooth_blend
 6. leader_orient     16. left
 7. continuous        17. field
 8. flat              18. all_around
 9. backing           19. tail
10. back_size         20. reference
```
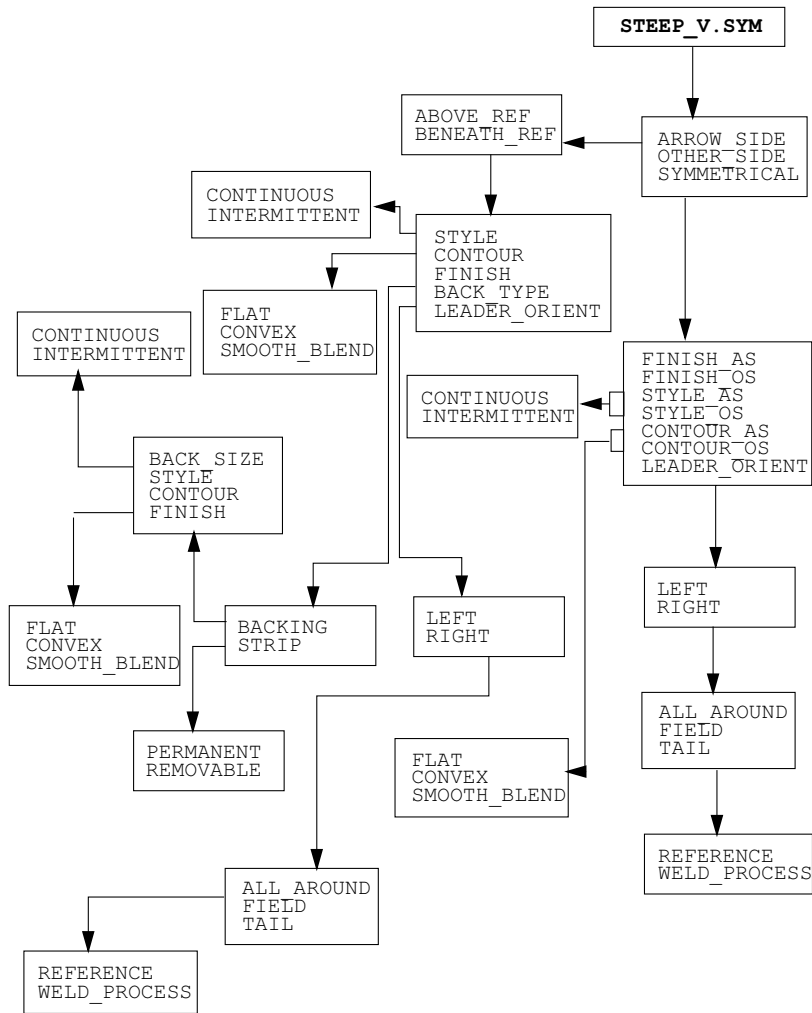
EXAMPLE 3

PICKS

 1. symmetrical   11. intermittent
 2. weld_size_as   12. continuous
 3. weld_size_os   13. convex
 4. style_as   14. flat
 5. style_os   15. left
 6. contour_as   16. all_around
 7. contour_os   17. field
 8. finish_as
 9. finish_os
10. leader_orient

8   I0x50(I00)
6   300

Symbol Definition Structure

BROAD_ROOT_V.SYM

ABOVE_REF
BENEATH_REF

ARROW_SIDE
OTHER_SIDE
SYMMETRICAL

CONTINUOUS
INTERMITTENT

WELD_SIZE
STYLE
CONTOUR
BACK_TYPE
FINISH
LEADER_ORIENT

FLAT
CONVEX
SMOOTH_BLEND

CONTINUOUS
INTERMITTENT

WELD_SIZE_AS
WELD_SIZE_OS
STYLE_AS
STYLE_OS
CONTOUR_AS
CONTOUR_OS
FINISH_AS
FINISH_OS
LEADER_ORIENT

CONTINUOUS
INTERMITTENT

BACK_SIZE
STYLE
CONTOUR
FINISH

LEFT
RIGHT

LEFT
RIGHT

FLAT
CONVEX
SMOOTH_BLEND

BACKING
STRIP

ALL_AROUND
FIELD
TAIL

PERMANENT
REMOVABLE

FLAT
CONVEX
SMOOTH_BLEND

REFERENCE
WELD_PROCESS

ALL_AROUND
FIELD
TAIL

REFERENCE
WELD_PROCESS

Fold Joint Symbol: Fold_Joint.sym

EXAMPLE 1

PICKS

```
1. arrow_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. weld_process
```

5.5   SDTR45

EXAMPLE 2

PICKS

```
1. other_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference
```

4   FGP167

Symbol Definition Structure

```
┌─────────────────────┐
│  FOLD_JOINT.SYM     │
└─────────────────────┘
            │
            ▼
       ┌──────────────────┐
       │ ARROW_SIDE       │
       │ OTHER_SIDE       │
       └──────────────────┘
            │
            ▼
    ┌──────────────────┐
    │ WELD_SIZE        │
    │ LEADER_ORIENT    │
    └──────────────────┘
            │
            ▼
    ┌──────────────┐
    │ LEFT         │
    │ RIGHT        │
    └──────────────┘
            │
            ▼
    ┌──────────────┐
    │ ALL_AROUND   │
    │ FIELD        │
    │ TAIL         │
    └──────────────┘
            │
            ▼
    ┌──────────────────┐
    │ REFERENCE        │
    │ WELD_PROCESS     │
    └──────────────────┘
```
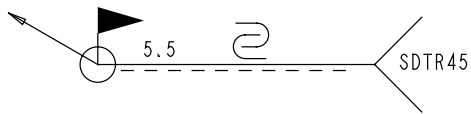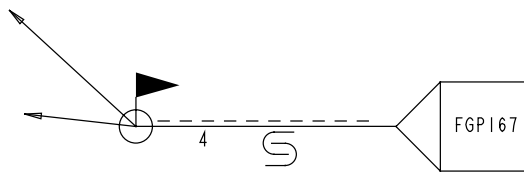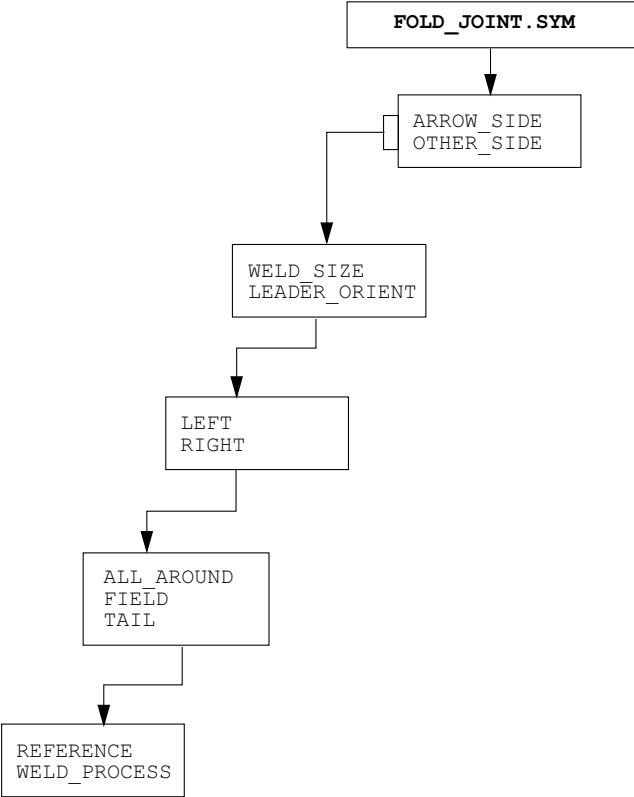
Combine Symbol: Combine.sym

EXAMPLE 1

```
PICKS

 1. as_above_ref    11. machine_as
 2. weld_type_as    12. concave
 3. weld_type_os    13. square_os
 4. weld_size_as    14. contour_os
 5. weld_size_os    15. flat
 6. weld_length_as  16. continuous
 7. weld_length_os  17. intermittent
 8. leader_orient   18. left
 9. u_weld_as       19. all_around
10. contour_as      20. field
                    21. tail
                    22. weld_process
```
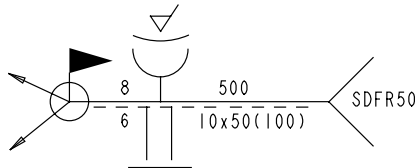

EXAMPLE 2

```
PICKS

 1. os_above_ref    11. machine_as
 2. weld_type_as    12. convex
 3. weld_type_os    13. fillet_os
 4. weld_size_as    14. contour_os
 5. weld_size_os    15. smooth_blend
 6. weld_length_as  16. intermittent
 7. weld_length_os  17. continuous
 8. leader_orient   18. left
 9. bevel_broad_as  19. field
10. contour_as      20. tail
                    21. reference
```

Symbol Definition Structure

```
CONTOUR_AS              COMBINE.SYM                CONTOUR_AS
MACHINE_AS                                         MACHINE_AS
                      AS_ABOVE_REF
                      OS_ABOVE_REF
FLAT                                               FLAT
CONVEX                                             CONVEX
CONCAVE                                            CONCAVE
SMOOTH_BLEND                                       SMOOTH_BLEND

              SQUARE_AS              SQUARE_AS
              V_WELD_AS              V_WELD_AS
              BEVEL_AS               BEVEL_AS
              U_WELD_AS              U_WELD_AS
              J_WELD_AS              J_WELD_AS
              V_BROAD_AS             V_BROAD_AS
              BEVEL_BROAD_AS         BEVEL_BROAD_AS
              FILLET_AS              FILLET_AS

CONTINUOUS                                         CONTINUOUS
INTERMITTENT                                       INTERMITTENT

              WELD_TYPE_AS           WELD_TYPE_AS
              WELD_TYPE_OS           WELD_TYPE_OS
              WELD_SIZE_AS           WELD_SIZE_AS
              WELD_SIZE_OS           WELD_SIZE_OS
              WELD_LENGTH_AS         WELD_LENGTH_AS
              WELD_LENGTH_OS         WELD_LENGTH_OS
              LEADER_ORIENT          LEADER_ORIENT

              LEFT              LEFT
              RIGHT             RIGHT

REFERENCE     ALL_AROUND        ALL_AROUND     REFERENCE
WELD_PROCESS  FIELD             FIELD          WELD_PROCESS
              TAIL              TAIL

CONTOUR_AS    SQUARE_OS         SQUARE_OS      CONTOUR_AS
MACHINE_AS    V_WELD_OS         V_WELD_OS      MACHINE_AS
              BEVEL_OS          BEVEL_OS
              U_WELD_OS         U_WELD_OS
FLAT          J_WELD_OS         J_WELD_OS      FLAT
CONVEX        V_BROAD_OS        V_BROAD_OS     CONVEX
CONCAVE       BEVEL_BROAD_OS    BEVEL_BROAD_OS CONCAVE
SMOOTH_BLEND  FILLET_OS         FILLET_OS      SMOOTH_BLEND
```
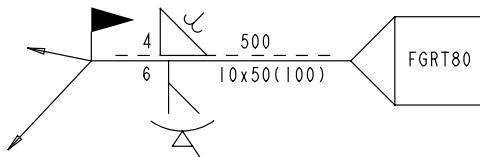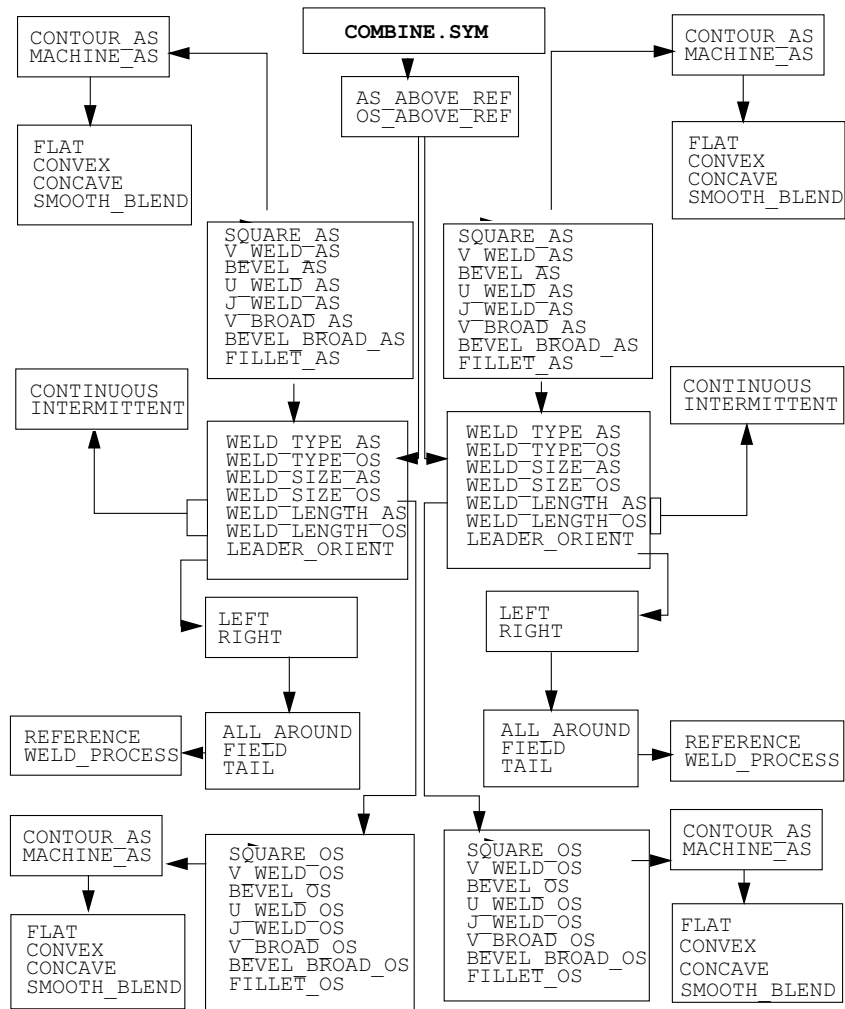
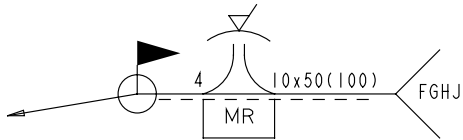## Edge Flanged Symbol: Iso_Edge_Flange.sym

EXAMPLE 1

PICKS
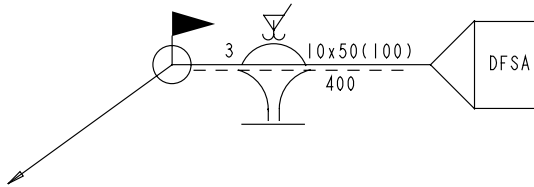
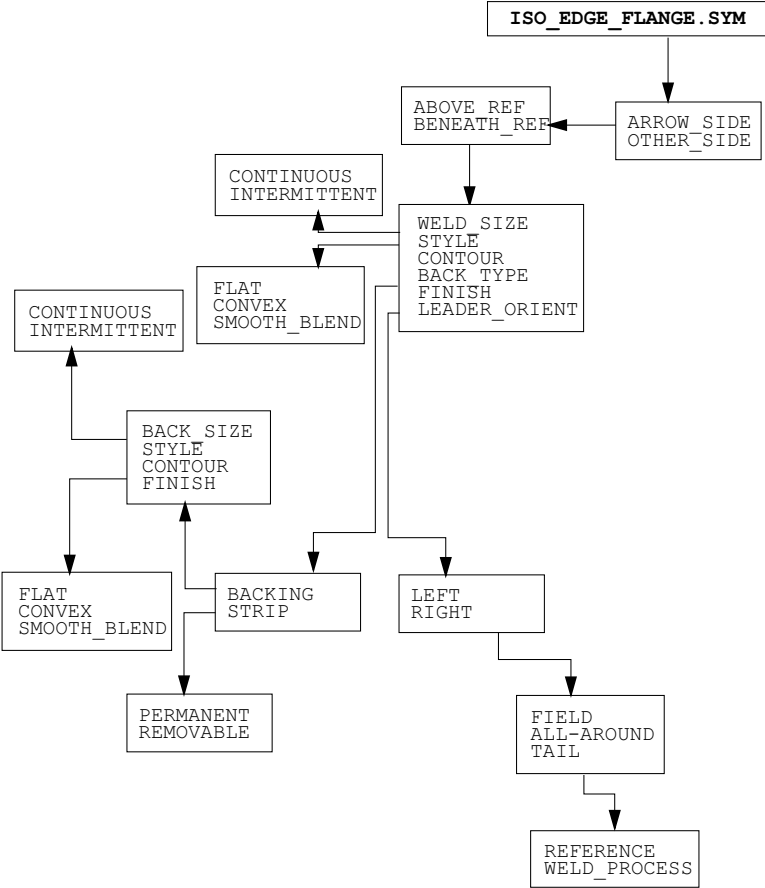|	
1. arrow_side	11. strip
2. above_ref	12. left
3. weld_size	13. removable
4. style	14. field
5. contour	15. all_around
6. back_type	16. tail
7. finish	17. weld_process
8. leader_orient	
9. intermittent	
10. convex	

EXAMPLE 2

PICKS

|	
1. other_side	11. style
2. beneath_ref	12. contour
3. style	13. finish
4. contour	14. intermittent
5. back_type	15. smooth_blend
6. leader_orient	16. left
7. continuous	17. all_around
8. flat	18. field
9. backing	19. tail
10. back_size	20. reference

Symbol Definition Structure

```
                                              ┌────────────────────────┐
                                              │ ISO_EDGE_FLANGE.SYM     │
                                              └────────────────────────┘
                                                          │
                                                          ▼
                           ┌──────────────┐      ┌──────────────┐
                           │ ABOVE_REF    │◄─────│ ARROW_SIDE   │
                           │ BENEATH_REF  │      │ OTHER_SIDE   │
                           └──────────────┘      └──────────────┘
                                  │
          ┌──────────────┐        ▼
          │ CONTINUOUS   │     ┌──────────────────┐
          │ INTERMITTENT │◄───►│ WELD_SIZE        │
          └──────────────┘     │ STYLE            │
                               │ CONTOUR          │
          ┌──────────────┐     │ BACK_TYPE        │
          │ FLAT         │     │ FINISH           │
          │ CONVEX       │     │ LEADER_ORIENT    │
          │ SMOOTH_BLEND │     └──────────────────┘
          └──────────────┘

  ┌──────────────┐
  │ CONTINUOUS   │
  │ INTERMITTENT │
  └──────────────┘
         ▲
         │      ┌──────────────┐
         │      │ BACK_SIZE    │
         └──────│ STYLE        │
                │ CONTOUR      │
                │ FINISH       │
                └──────────────┘
                     ▲
  ┌──────────────┐   │   ┌──────────────┐     ┌──────────────┐
  │ FLAT         │   │   │ BACKING      │     │ LEFT         │
  │ CONVEX       │   │   │ STRIP        │     │ RIGHT        │
  │ SMOOTH_BLEND │   │   └──────────────┘     └──────────────┘
  └──────────────┘   │                               │
                     ▼                               ▼
              ┌──────────────┐              ┌──────────────┐
              │ PERMANENT    │              │ FIELD        │
              │ REMOVABLE    │              │ ALL-AROUND   │
              └──────────────┘              │ TAIL         │
                                            └──────────────┘
                                                   │
                                                   ▼
                                            ┌──────────────┐
                                            │ REFERENCE    │
                                            │ WELD_PROCESS │
                                            └──────────────┘
```

Edge Symbol: Edge_Weld.sym

EXAMPLE 1

```
PICKS

1. arrow_side        11. all_around
2. weld_size         12. tail
3. style             13. weld_process
4. contour
5. finish
6. leader_orient
7. intermittent
8. convex
9. left
10. field
```
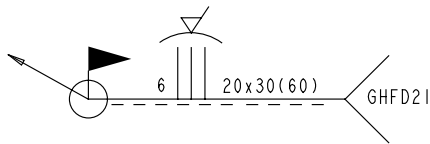

EXAMPLE 2

```
PICKS

1. other_side        11. reference
2. style
3. contour
4. leader_orient
5. continuous
6. smooth_blend
7. left
8. field
9. all_around
10. tail
```

Symbol Definition Structure

```
                                        ┌──────────────────┐
                                        │  EDGE_WELD.SYM   │
                                        └──────────────────┘
                                                 │
                                                 ▼
                                        ┌──────────────────┐
                                        │  ARROW_SIDE      │
                                        │  OTHER_SIDE      │
                                        └──────────────────┘
       ┌──────────────────┐            ┌──────────────────┐
       │  CONTINUOUS      │            │  WELD_SIZE       │
       │  INTERMITTENT    │            │  STYLE           │
       └──────────────────┘            │  CONTOUR         │
                                       │  BACK_TYPE       │
       ┌──────────────────┐            │  FINISH          │
       │  FLAT            │            │  LEADER_ORIENT   │
       │  CONVEX          │            └──────────────────┘
       │  SMOOTH_BLEND    │
       └──────────────────┘            ┌──────────────────┐
                                       │  LEFT            │
                                       │  RIGHT           │
                                       └──────────────────┘

                           ┌──────────────────┐
                           │  ALL_AROUND      │
                           │  FIELD           │
                           │  TAIL            │
                           └──────────────────┘

                ┌──────────────────┐
                │  REFERENCE       │
                │  WELD_PROCESS    │
                └──────────────────┘
```
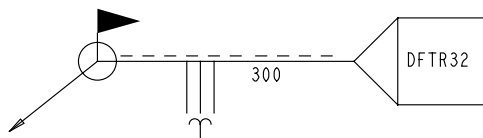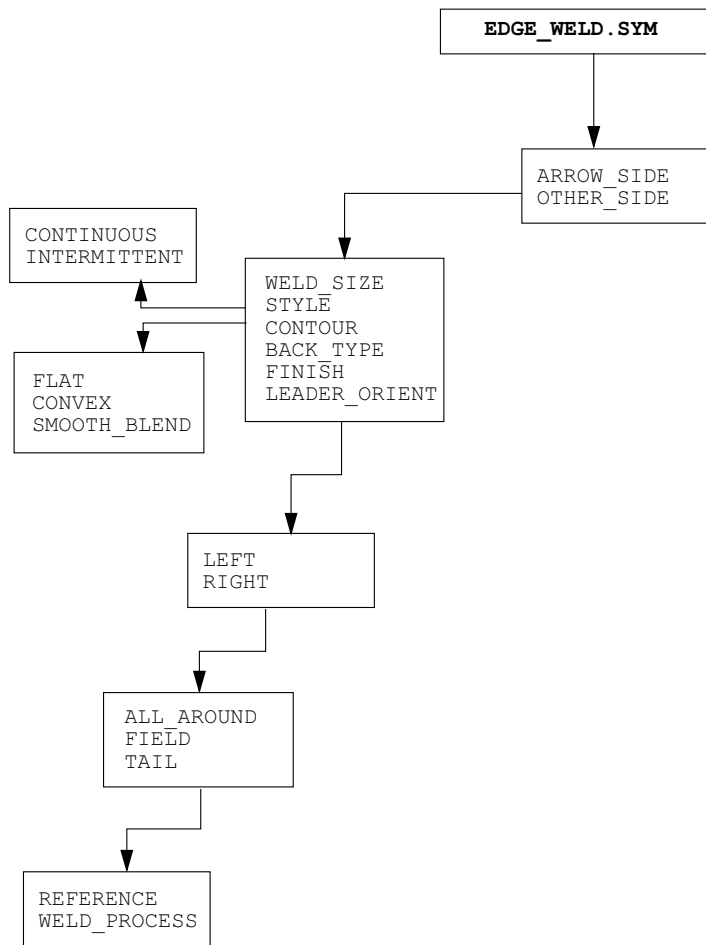
About the Symbol Instance Palette

The Symbol Instance palette is a customized drawing that contains symbols created and manually added by you. Symbol definition attributes such as height, angle, grouping, variable text value, and so forth can be used for each specific symbol instance created. However, these attributes can be modified only while creating the symbol instance and not within the customized symbol instance palette.

The Symbol Instance palette file is stored in a location specified by the following configuration option:

`symbol_instance_palette_file`

This option's value is the directory and file name used to store the palette and where the palette is retrieved between sections.

You can perform the following functions using the Symbol Instance dialog box:

- Select a symbol instance in the palette and place it in a drawing.

- Add a symbol drawing to the Symbol Instance palette.

- Arrange symbols in the Symbol Instance palette.

To Select a Symbol Instance in the Palette

You can select a symbol instance in the **Symbol Instance Palette** and place it in a drawing. The **Pick in Palette** button, located in the **Symbol Instance** dialog box, opens the **Symbol Instance Palette** dialog box that contains the same characteristics as a typical drawing window. Therefore, zooming, panning, and other functions can be applied.

1. Click **Insert > Symbol Instance**. The **Symbol Instance** dialog box opens.
2. Click **Pick in Palette**. The **Symbol Instance Palette** dialog box opens. Click the symbol that you want to insert in the drawing. The symbol is displayed in the preview area and the **GET POINT** menu appears.
3. Place the symbol instance in the drawing by clicking on the desired location.

Note: Pick in Palette is a quick way of selecting only one symbol. If a symbol is selected in the palette and then cancelled by selecting **Quit** from the **GET POINT** menu, you cannot select a different symbol. You must click **Pick in Palette** again to select another symbol.

To Add a Drawing Symbol to the Palette

You can create or obtain a drawing symbol and then add it to the Symbol Instance palette.
1. Click **Insert > Symbol Instance**. The **Symbol Instance** dialog box opens.
2. Click **Add to Palette**. The **Symbol Instance Palette** dialog box opens and the **GET POINT** menu appears. The symbol shown in the symbol definition field appears in the **Symbol Instance Palette**. If the **Symbol Instance Palette** dialog box is not already opened, the message, `Select Symbol Location` appears.

Note: The **ATTACH TYPE** menu may appear, depending on the active symbol attributes.

3. Place the symbol in the palette by clicking on the desired location.

To Arrange and Delete Symbols in the Palette

You can arrange and delete symbols in the Symbol Instance palette by using the **Move Symbol** and **Delete Symbol** buttons.

To move a symbol within the symbol instance palette:

1. Click **Move Symbol** and then click the symbol that you want to move.
2. Relocate the symbol anywhere within the Symbol Instance palette. Use the middle mouse button to cancel move activity.

To delete a symbol within the Symbol Instance palette, click **Delete Symbol** and then click the symbol that you want to delete.

The system prompts you to confirm or reject changes when you close the Symbol Instance palette.

To Create a User-Defined Weld Symbol

Redefine an existing system weld symbol by doing one or all of the following:

- Add as many copies of variable texts as you want.
- Change the default values of variable texts.
- Add and delete as many notes and entities as you want, and place new ones in any group (or in no group at all).
- Redefine the cosmetics of existing notes and entities.
1. Save the symbol by assigning a new temporary name.
2. Move the original (system-supplied) symbol from the system weld library to another directory or rename it. The system weld symbol libraries are located in the installation directory path `<install_dir>/symbols/library_syms/weldsymlib`.

3. Move the new user-defined symbol into the system weld library and assign it the same name as the original.
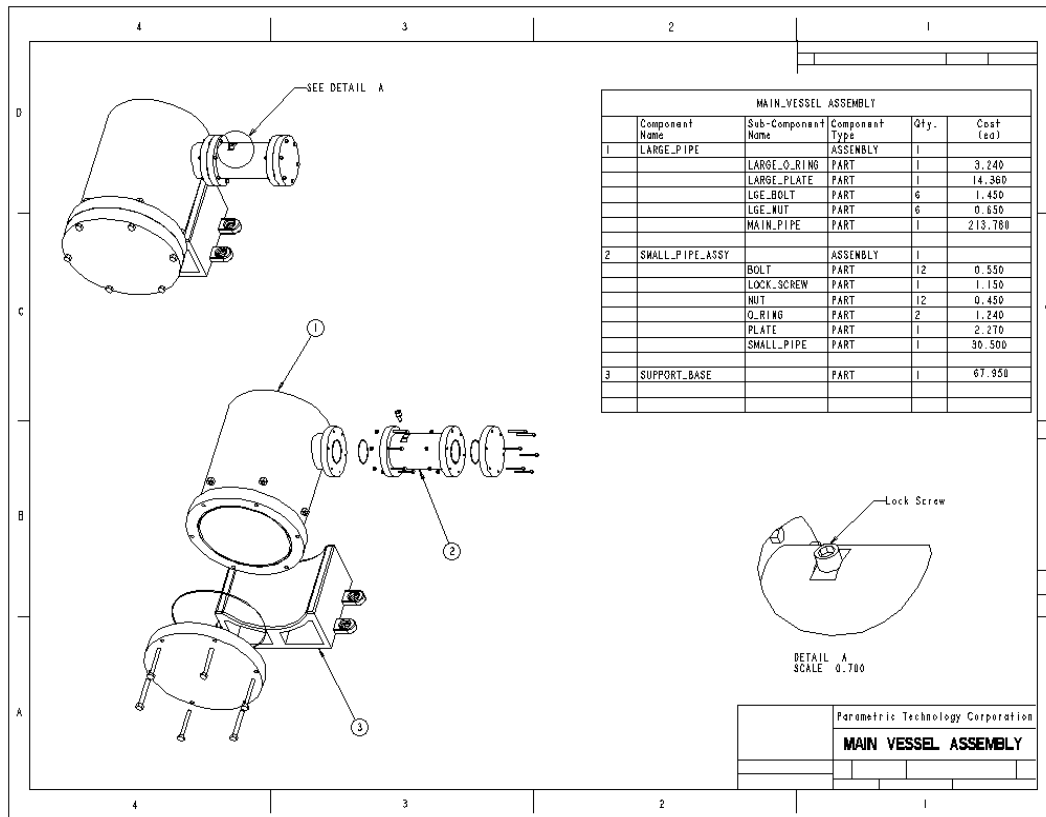
About Creating Reports

In Drawing mode, you can use Pro/REPORT to create dynamic, customized reports with drawing views and graphics. Pro/REPORT also works with Pro/HARNESS-MFG, Pro/PIPING, Pro/MFG, and Pro/DIAGRAM. By defining repeat regions, you can add model data to a report, which expands and shrinks as the model changes. You can also use the following techniques to manage the data that appears in a report:

• Add filters to eliminate specific types of data from appearing in reports, drawing tables, or layout tables.

• Search recursive or top-level assembly data for display.

• List duplicate occurrences of model data individually or as a group in a report, drawing table, or layout table.

• Directly link assembly component balloons to a customized BOM and automatically update them when you make assembly modifications.

You can generate several kinds of output using Pro/REPORT such as family tables, associative reports, and graphical wirelists. A common example of a report is a customized Bill of Materials (BOM), such as the one shown in the next figure.

Report with Assembly Views

Creating Reports

In Report mode, you can display data in tabular form in reports just as it is in drawing tables. The system takes the data that it reports in the tables directly from a selected model and updates it when you change the model. However, you cannot modify all model dimension values and geometry in Report mode—they are *read-only*. The system gives report files the file extension .rep.

Report Mode Menus

Report mode menus are abbreviated versions of the Drawing mode menus. This mode provides all of the associative abilities that Drawing mode offers, except that you cannot modify the model. Refer to the appropriate chapters of this guide for additional information on drawing commands.

To Create a Report

1. From the Pro/ENGINEER menu bar, choose **File** > **New...**.
2. In the New dialog box, click **Report** and type a name in the **Name** box (or accept the default); then click **OK**.
3. In the **Create Report** dialog box, specify the report size or retrieve a format:

- To specify the size, select **Set Size** and do one of the following:

  – Click **Portrait** in the **Orientation** box (to make the height larger than the width) and select a standard size from the **Standard Size** list.

  – Click **Landscape** in the **Orientation** box (to make the width larger than the height) and select a standard size from the **Standard Size** list.

  – Click **Variable** in the **Orientation** box to define both the height and width dimensions. Select **Inches** or **Millimeters** and type values in the **Width** and **Height** boxes.

- To retrieve a format, click **Retrieve Format** and select a name from the **Name** list in the **Format** box. You can also type [?] or click **Browse** to select a name from the Open dialog box.

4. Click **OK**. The system displays the report as specified and the REPORT menu appears.
5. Create a drawing table on the drawing sheet by choosing REPORT > **Table** > **Create.**
6. Define a repeat region by choosing TABLE > **Repeat Region** > **Add.**
7. Select cells to repeat with model information.

Creating a Repeat Region
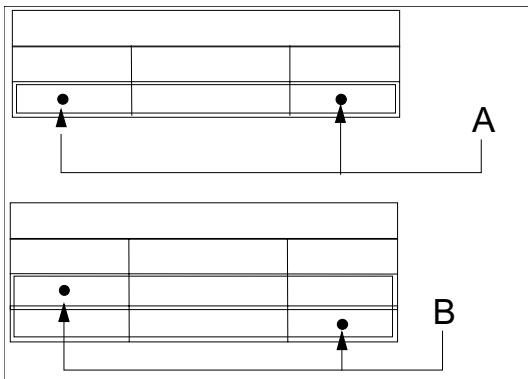
Selecting A yields highlighted (one-row) repeat region

Selecting B yields highlighted (two-row) repeat region

8. Choose **Enter Text** and type title text in a row or column outside of the repeat region (unless you want that text to repeat with every occurrence of model data).
9. The repeat region in the table must contain symbol parameter information to be shown in the table. Choose TABLE > **Enter Text** > **Keyboard**; or choose ENTER CELL > **Report Sym**.
10. Add the assembly model to the report by choosing REPORT > **Views** and typing the name of the assembly.
11. The system automatically chooses **Add View** from the VIEWS menu, and the VIEW TYPE menu appears. To add views of the assembly to the report, add a general view to the sheet. If you do not want views of the model in the report, choose
VIEW TYPE > **Quit** and VIEWS > **Done/Return**. The system still adds the model to the report, but it does not display it on the sheet. You can add drawing views at any time.
12. If necessary, choose **Repaint** to display the contents of the table.
13. Add a drawing format to the report or change it by choosing SHEETS > **Format** > **Add/Replace**. You can predefine tables on these drawing formats, save them, and recall them at any time into a report or drawing.

You may also add the drawing table used in the report by including it in a drawing format that you can retrieve. When you add a drawing format containing a table to a report, drawing, or layout, the table becomes independent of the format. If you decide to replace the format, the system highlights the table, and you can delete it.

Including Report Parameters in a Repeat Region

Pro/ENGINEER extracts values for parameters from model data and displays them in tables when the parameters are included within a repeat region. These parameters can be user-defined or they can already be defined by Pro/REPORT.

Pro/REPORT Parameters

PARAMETER NAME	DEFINITION
&asm.mbr.cparams.name	Lists the names of all user-defined parameters in an assembly component. This parameter is defined for the parts making up the connector outside of the cabling environment.
&asm.mbr.cparams.value	Lists the values of all user-defined parameters in an assembly component. This parameter is defined for the parts making up the connector outside of the cabling environment.
&asm.mbr.cparam.User Defined	Lists the specified user-defined parameters used in an assembly component. These parameters are defined for the parts making up the Connector outside of the cabling environment.
&asm.mbr.name	Displays the name of an assembly member. To show tie wraps and markers, the region attributes must be set to **Cable Info**.
&asm.mbr.param.name	Lists the names of all user-defined parameters in an assembly member.
&asm.mbr.param.value	Lists the values of all user-defined parameters in an assembly member.
&asm.mbr.type	Displays the type (part or assembly) of an assembly member.
&asm.mbr.User Defined	Lists the specified user-defined parameter for the respective assembly components. Note that "&asm.mbr." can be used as a prefix before any User-defined parameter in an assembly member.
&fam.inst.name	Displays the name of a family table instance.
&fam.inst.param	Displays the dimensions of suppressed features in a 2-D repeat region as dashes.

	This is not a complete symbol; only suppresses dashed dimensions if the drawing setup file option "dash_supp_dims_on_region" is set to "yes," the default.
&fam.inst.param.id	Displays the ID of a family table parameter if it is a dimension.
&fam.inst.param.name	Displays the name of a family table parameter.
&fam.inst.param.value	Displays the value of a family table parameter for an instance.
&lay.param.name	Lists the names of all user-defined parameters in a layout.
&lay.param.value	Lists the values of all user-defined parameters in a layout.
&mdl.param.name	Lists the names of all user-defined parameters in a model.
&mdl.param.value	Lists the values of all user-defined parameters in a model.
&rpt.index	Displays the number assigned to each record in a repeat region.
&rpt.level	Shows the recursive depth of an item.
&rpt.qty	Displays the quantity of an item.

Assembly Component Parameters

The following system parameters exist for parameters that you enter into an assembly as feature relations and those that exist at the assembly level only:

- "&asm.mbr.cparams.name"
- "&asm.mbr.-cparams.value"
- "asm.mbr.cparam.parametername"

Note: These parameters do not exist at the part level as "asm.mbr.parametername" and other parameters can.

In the next example, subassembly "LARGE_PIPE" is assigned three parameters:

- "price," a number parameter
- "availability," a string parameter
- "in_house,' a "yes_no" parameter

The part "LARGE_PLATE" is given the "price" parameter as well. In subassembly "SMALL_PIPE," one occurrence of part "O_RING" is assigned the "price" parameter also; the parameter exists in assembly "SMALL_PIPE" and not in part "O_RING." When the system parses the component parameters for the top-level assembly "Main Vessel," it includes the parameter. Every parameter that is assigned to a component in this example is added to its parent assembly as a feature relation. This figure shows the repeat region template using the component parameters.

Repeat Region Template Using the Component Parameters

MEMBER NAME	MEMBER TYPE	PRICE	PARAMETER	VALUE
asm.mbr.name	asm.mbr.type	asm.mbr.cparam.price	asm.mbr.cparams.name	asm.mbr.cparams.value

About Using Pro/REPORT with Pro/MFG

You can use Pro/REPORT with Pro/MFG to access manufacturing parameters at various levels and create customized reports on the manufacturing process to suit your specific needs.

About Using Pro/REPORT in a Piping Assembly

You can use Pro/REPORT functionality with Pro/PIPING to tailor the piping bend machine and bend location information to suit your required format by switching columns, subtracting a constant, and/or changing the sign of the values.

About Repeat Regions
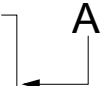
Repeat regions are user-defined rows and columns, or combinations of rows and columns (cells) that duplicate themselves to accommodate the amount of data that the model currently possesses. They contain the following:

- System and user-defined parameters for which the values are extracted from the model that is associated with the report
- Standard table text

By using repeat regions, your tables containing report data can expand and contract with varying quantities of data supplied by the models. The following example shows a BOM with repeat regions.

BOM with Repeat Regions

Top-Level Assembly BOM		
Index	Component Name	Quantity
1	LARGE_PIPE	1
2	SMALL_PIPE_ASSY	1
3	SUPPORT_BASE	1

A

A - Expanded Repeat Region
If you have a license for Pro/REPORT, you can create a repeat region in any mode in which you can create a table (Drawing, Report, Layout, Diagram, or Format mode). You can nest repeat regions or make them two-directional.

> **Note:** You can specify multiple repeat regions for a single table, but they cannot overlap.

About Using Assembly Simplified Representations

You can use assembly simplified representations in a drawing to control a repeat region of the model, and you can also show BOM balloons on the representation. When you use assembly simplified representations to control a repeat region, simplified representations of the same assembly behave entirely as different models.

A repeat region that reports on a simplified representation of a model reports on information within that simplified representation, as opposed to reporting on the assembly itself. This includes the following:

- If you substitute a component with anything other than a simplified representation of itself, the assembly tree itself is different.
- The system does not show excluded components, even if they are in memory.
- Parameter values set by assembly relations (except mass properties) reflect the represented states of the models within the assembly.
- A repeat region is automatically associated with the current model or simplified representation.
  - If the current simplified representation changes, the repeat region is always associated with the original simplified representation when it was created or loaded.

- If the current model is an assembly, the repeat region is associated with that assembly model.
- If the current model is a simplified representation, it is associated with the simplified representation.

- If you substitute a part or subassembly with a simplified representation, the system still displays it in the correct location in the assembly tree.

  For example, if you substitute a part in the third level of the assembly with a simplified representation of the part, the report displays the substitution in the third level. This applies only to substitutions by simplified representations.

You can create a table or Bill of Materials with piping information in a drawing, so that the system updates the table automatically to reflect changes in the piping assembly. You can also customize drawing tables to reflect practices specific to your particular company.

Naming Conventions for Simplified Representations in a Repeat Region

The naming conventions for simplified representations in a repeat region are as follows:

- If you substitute a component with a simplified representation, the system gives it the original part name.
- If you substitute a component with a family table instance, the system gives it the instance name.
- If you substitute a component with an envelope, the system gives it the envelope component name.
- If you substitute a component with a geometry snapshot, the system gives it the original part name.
- If you substitute a component with an interchange group, the system gives it the interchange part or assembly (and part) names.

Restrictions When Using an Assembly Simplified Rep to Drive a Repeat Region

The following restrictions apply when you use an assembly simplified representation to drive a repeat region:

- Pro/REPORT supports designated parameters of geometry snapshots only in repeat regions, not in notes.
- Component parameters are not available for substituted parts.
- The system supports the following report symbols for simplified representation models:
  - asm (including cable)
  - harn
  - rel
  - mdl
  - rpt
  - weld.asm
  - piping

In the following figure, a table has been added to a report. A model has not been added to the report yet, so data is not listed in the columns. In the next figure, an assembly model has been added to the report that contains the two subassemblies, "Large Pipe" and "Small Pipe," so the repeat region grows to accommodate the requested data.

Table Created on a Report

before update for model data →	Top-Level Assembly BOM		
	Index	Component Name	Quantity
repeat region row →			

Growing Repeat Region

report table
after model is
added

Top-Level Assembly BOM		
Index	Component Name	Quantity
1	LARGE_PIPE	1
2	SMALL_PIPE_ASSY	1

expanded repeat region

If you add an additional component to the assembly, the system updates the repeat region accordingly. In the following figure, the part "Support Base" was added to the same assembly and after updating the table, the repeat region grows to accommodate the new component member.

Part Added to Assembly

Top-Level Assembly BOM		
Index	Component Name	Quantity
1	LARGE_PIPE	1
2	SMALL_PIPE_ASSY	1
3	SUPPORT_BASE	1

expanded repeat region

Defining Repeat Regions

Throughout most of this chapter, an assembly named "Main Vessel" is used to illustrate the properties of repeat regions. The following figure illustrates the composition of this assembly.

Sample Assembly Structure
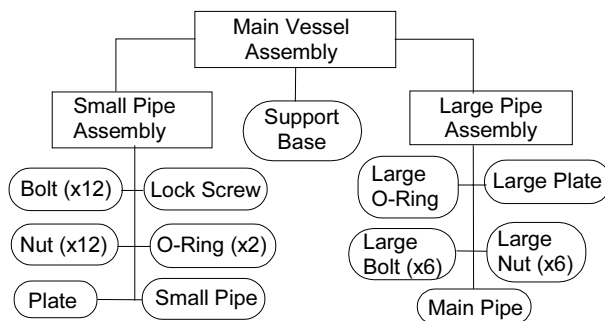
If you set the drawing setup file option "model_digits_in_region" to "yes," two-dimensional repeat regions reflect the number of digits of the part and assembly model dimensions. To change the number of digits of the dimension in the drawing, you must show the model dimensions on the drawing and use the **Num Digits** command.

To Add a Simple Repeat Region to a Table

1. Choose TBL REGIONS > **Add** > **Simple**.
2. Select the uppermost row of the repeat region.
3. Select the bottom row of the repeat region.

To Remove a Repeat Region from a Table

1. Choose TBL REGIONS > **Remove**.
2. Do one of the following:
• Choose RMV REGION > **Pick Region.**
  – If the table is in *numeric* mode, the system *freezes* the region's contents into nonparametric text. If other regions exist in the table, it repaginates them (with page breaks only possible inside the remaining

regions). Otherwise, it appends all segments to the first, and strips pagination information.

- If the table is in *symbolic* mode, it shrinks the region to its template and then removes it (this returns the table to its precise state just before creating the region).

- Choose RMV REGION > **All Regions**. If the table has multiple segments (and is in numeric mode), the system asks you if you want to explode all paginated table segments into separate tables. If you confirm, it creates separate tables for each existing segment. Otherwise, it concatenates all segments into one big table, and strips the pagination information.

To Display a Different Model

Using the **Model / Rep** command in the TBL REGIONS menu, you can display a different model or simplified representation to control a region. If you change the model associated with the repeat region, the current model appears as the default choice.

1. Choose TBL REGIONS > **Model / Rep**.
2. Select a region and type a new model name. For example: ASM_1.ASM.
3. Choose **Confirm** to delete BOM Balloons and model-dependent region items.

To Create a Table with No Duplicates and with Recursion

1. Choose REPORT > **Table** > **Repeat Region** > **Attributes**.
2. Select within the repeat region.
3. Using the REGION ATTR menu, do one of the following:
- Choose **Recursive** and search all levels of the assembly.

- Choose **No Duplicates** and merge all occurrences of the same member into one line, accompanied by a value for quantity in the right column. This affects your BOM balloon display. BOM balloons related to instances with the same parameters merge the balloons into one.

Example: Table After Choosing No Duplicates and Recursive

INDEX	NAME	QTY
1	BOLT	12
2	LARGE_O_RING	1
3	LARGE_PIPE	1
4	LARGE_PLATE	1
5	LGE_BOLT	6
6	LGE_NUT	6
7	LOCK_SCREW	1
8	MAIN_PIPE	1
9	MAIN_VESSEL	1
10	NUT	12
11	O_RING	2
12	PLATE	1
13	SMALL_PIPE	1
14	SMALL_PIPE_ASSY	1
15	SUPPORT_BASE	1

To Enter Report Parameters into a Table

If one or more repeat regions already exist, you can type report parameters into a table manually or by choosing them from a menu.
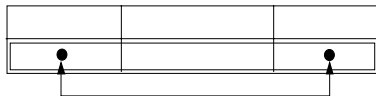
1. Choose ENTER CELL > **Report Sym**.

2. Select the cell of the table in which you want to type the text. The REPORT SYM menu displays commands for the first parameter elements: **asm**, **dgm**, **fam**, **harn**, and **rpt**.

3. Choose an element. If it needs another, the REPORT SYM menu appears again, with commands for the next parameter element. You can use **UP** to navigate through the tree structure. Whenever an element needs another element, the system follows the element name by ellipses (...).

The next figure presents a report with a table consisting of three columns and two rows.

A repeat region contains the cells from the entire lower row.

Repeat Region with Cells from Entire Lower Row

those two cells selected to
create repeat region

In the next figure, the column header text is typed. In this example, an index, the assembly member name, and the quantity information are intended to be shown.

Repeat Region with Column Header Text

Index	Component Name	Quantity

The report parameter symbols "&rpt.index," "&asm.mbr.name," and "&rpt.qty" are typed into the three cells that make up the repeat region. The system automatically includes an ampersand preceding all parameter symbols if you enter the text by choosing **Report Sym** from the ENTER CELL menu. The result appears in the following figure.

Repeat Region with Report Parameter Symbols

Index	Component Name	Quantity
&rpt.index	&asm.mbr.name	&rpt.qty

To add the top-level assembly, "Main Vessel," to the report, choose **Views**, and then type [`Main Vessel`]. You do not have to place the drawing view of the assembly at this time. The table expands after you select the model. All of the default attributes are chosen for this repeat region (that is, duplicates, flat). The quantity values are not listed in this expanded table because the **Duplicates** command lists all occurrences of assembly members separately. Therefore, all quantities are assumed to be 1. The resulting table appears as shown in the following figure.

Report with Top-Level Assembly

Index	Component Name	Quantity
1	LARGE_PIPE	
2	SMALL_PIPE_ASSY	
3	SUPPORT_BASE	

Tip: Assigning Feature Relation Parameters

When you assign feature relation parameters to assemblies, you *must* select the assembly component, not a feature of the component. For example, when you query-select to assign the parameter, the message area reads "Showing partname component No. X in assemblyname," instead of "Showing feature X (feature type) in part partname." If you select the feature inadvertently, the assigned parameter values do not parse on the report because the report symbols "&asm.mbr.cparam(s)" search only for assembly parameters. When the system

parses the symbols, the table looks like the example in the following figure.

The repeat region has attributes of **Duplicates** and **Recursive** specified. The column with the heading "PRICE" lists only values for the parameter symbol "price." The column with the heading "PARAMETER" lists the names of all of the user-defined parameters assigned to each component. The column with the heading "VALUE" lists the values of all of the user-defined parameters assigned to each component. These parameters are sorted in the order in which they were added to the model. Note that where a component has multiple parameters (assembly "Large Pipe"), multiple entries are present. (You might want to use a nested repeat region in this case to differentiate the multiple entries from the other assembly components.)

Table with Parameter Symbols Parsed

MAIN_PIPE	PART	105.500	PRICE	105.500
MAIN_PIPE	PART	105.500	AVAILABILITY	07 Jul 93
MAIN_PIPE	PART	105.500	IN_HOUSE	False
LARGE_O_RING	PART			
LARGE_PLATE	PART			
LGE_BOLT	PART			
LGE_BOLT	PART			

LGE_NUT	PART			
LGE_NUT	PART			
SUPPORT_BASE	PART	67.500	PRICE	67.500
SMALL_PIPE_ASSY	ASSEMBLY			
SMALL_PIPE	PART			
PLATE	PART			
O_RING	PART	0.450	PRICE	0.450
LOCK_SCREW	PART			
BOLT	PART			
BOLT	PART			

NUT	PART			
BOLT	PART			
BOLT	PART			
BOLT	PART			
BOLT	PART			
BOLT	PART			
BOLT	PART			
O_RING	PART			

Note: only one O-Ring assigned "price" parameter

Table is broken for display purposes so as not to show all patterned bolts and nuts

About Cabling Component Parameters

You can include cabling parameters in drawing and report tables using the "asm.mbr.cbprm.User Defined" parameter symbol in a repeat region. This parameter symbol displays any user-defined parameters used in cabling components. If you use the parameter symbol "asm.mbr.cbprms.Name/Value" in a repeat region, any pre-defined cabling parameter names and values (such as wire type units, name, and so on) appear.

When you include any of these parameter symbols in a repeat region, the commands **Cable Info** and **No Cbl Info** appear in the REGION ATTR menu. You must choose **Cable Info** to display any cabling information in the selected region. The **No Cbl Info** command displays only non-cabling assembly information in the table. If you choose **Cable Info** for a repeat region, the table expands to include all spool names, tie wraps, and markers that exist on the model when "asm.mbr.name" or "asm.mbr.type" parameter symbols are used in the repeat region. You cannot use repeat region attributes **No Duplicates** or **No Dup/Level** when you use the "asm.mbr.cblprms.name" parameter symbol in that region.

About Creating Pro/REPORT Tables in Flat Harnessed Drawings

Using the top-level report symbol, "mbr," you can create Pro/REPORT tables in a flat harnessed drawing for a single assembly component/connector. When you enter an "mbr" symbol such as "mbr.name" into a repeat region, the system arbitrarily selects a component from the default drawing model and reports only on that component, producing the relevant subset of the table that would be produced if you had typed the "asm.mbr" symbol instead.

To change the component that is driving the repeat region, you must display it in the view. When you choose **Model / Rep** from the TBL REGIONS menu and select a region driven by the symbols beginning with "mbr," the system prompts you to select a component in a view. When you select a component, that component then drives the repeat region.

About Showing Terminators in Report Tables

You can show terminators in report tables for cable assemblies that have connectors with terminator parameters by using the report symbols "&asm.mbr.name" and "&asm.mbr.type." To show the terminators, set the drawing setup file option "show_cbl_term_in_region" to "yes" (the attribute **Cable Info** must be set for the repeat region). When creating new drawings, the default value is "yes." For existing drawings, the default value is "no."

To Use Report Parameters in Multi-Model Drawings

To create tables that display data for different models, you must change the active model, and then create a new repeat region and type the parameters. Otherwise, the system takes data from the last active model.

Repeat regions always remember the models from which the system is extracting their values. A report parameter that you type in a repeat region on the drawing while a certain model is active reads values from that model only. The system extracts report parameter values only when you include the parameters within a repeat region. For example, if you type a set of report parameters in a table without a repeat region, the system does not read parameters from the active model until they are enclosed by a repeat region.

If the report parameter symbol "&asm.name" resides inside a table but outside of a repeat region, it can have a postfix number (for example, "&asm.name:3") to specify the assembly in a multi-model drawing from which it is reading its information. These postfix numbers always begin counting at "0."

To Enter Report Parameter Values into Empty Repeat Region Cells

You can type a user-defined parameter value in an empty repeat region cell for an assembly member that does not have that parameter value. If you type the parameter value in the table, the system automatically adds it to the corresponding assembly member.

You can use this method only for the report parameter symbol "asm.mbr.*parametername*," where *parametername* is a user-defined parameter, and the repeat region template must contain the parameter symbol "asm.mbr.*parametername*." That is, you specify the parameter value to appear in the table by typing the parameter name symbol, but the values do not exist in the models and consequently do not appear.

To Enter the Value of a User-Defined Parameter

1. Choose DETAIL > **Modify** > **Value** (or choose **Modify** from the LAYOUT or DIAGRAM menu).
2. Select an empty repeat region cell (empty of value only, the region has the parameter symbol). If it contains more than one parameter symbol that does not have a value, a menu displays the parameters in the region for which you can type a value.

3. Type the value for the parameter. The system adds the parameter value to the corresponding assembly member.
4. Choose **Update Table**, **Regenerate**, or **Repaint** to update the value of the parameter in other repeat regions (or any command that redisplays them).

Modifying Values

You can type the value of a user-defined parameter in Drawing, Layout, or Diagram mode, but not in Report mode. Report mode is *read-only* and modifications do not reflect back to the model database. However, you can modify the following parameter values in Report mode, and the value does reflect back to the model database:

- "&asm.mbr.parametername"
- "&asm.mbr.param.value"
- "&mdl.param.value"
- "&lay.param.value"

You can modify the value of these user-defined report parameters within repeat regions using the **Modify** and **Value** commands. Modifying these values on the sheet on which they appear changes them throughout the database. You can also modify them if they appear in notes.

To Modify Repeat Region Attributes

You can modify repeat region attributes in Format, Report, Drawing, and Layout modes. When you choose **Attributes** from the TBL REGIONS menu, the REGION ATTR menu displays the following commands:

- **Duplicates**—Lists duplicate occurrences of a parameter individually in a repeat region. This command sorts by feature number any data from the "&asm.mbr.name" parameter symbol display.
- **No Duplicates**—Lists duplicate occurrences of a parameter singly, along with a quantity if you use the parameter "&rpt.qty" in the region. This command sorts by parameter value any data from the "&asm.mbr.name" parameter symbol display.
- **No Dup/Level**—For a selected level of the region, lists duplicate occurrences of a parameter singly, along with a quantity if you use the parameter symbol "&rpt.qty" in the region. This command allows you to list duplicates only if the object resides in different levels of the assembly. There are no duplicates per level in the assembly. It sorts by parameter value any data generated from the "&asm.mbr.name" parameter symbol display.
- **Recursive**—Searches all levels of data for parameters.
- **Flat**—Searches only the top level of data for parameters.
- **Min Repeats**—Sets the minimum number of repetitions for a repeat region. The default minimum is "1." The system leaves extra rows blank. If the minimum is set to "0," it avoids blank lines caused by the lack of data.
- **Start Index**—Begins the index numbering of a repeat region (the value of "&rpt.index") where the index numbering of another repeat region ended. You cannot assign this attribute to nested repeat regions.
- **No Start Idx**—Begins the index numbering of a repeat region at 1.
- **Cable Info**—Displays the cable parameter information in tables containing appropriate parameter symbols.
- **No Cbl Info**—Displays only assembly parameter information in tables (that is, no cabling information appears).
- **Bln By Part**—When you suppress or replace a component to which a BOM balloon is attached, this command reattaches the balloon to another placement of the same part.
- **Bln By Comp**—Specifies that simple BOM balloons reattach themselves to whatever component replaced the one that originally owned the BOM balloon.

    Note: The *template* cell drives the number of digits of a quantity. In a two-directional repeat region, modifying the number of digits in the first column affects every column.

Example: Controlling Attributes

The next figure presents the same table as that shown in Entering Report Parameters into a Table on page 11 - 17. The repeat region contains the system parameter symbols "&rpt.index," "&asm.mbr.name," and "&rpt.qty" (for system parameters. The default repeat region attributes of **Duplicates** and **Flat** are used. In Report mode, the assembly "Main Vessel" is added to a report, and then the table is added to the report.

Repeat Region Containing System Parameters

Index	Component Name	Quantity
&rpt.index	&asm.mbr.name	&rpt.qty

The resulting table looks like the one in the following figure. The index numbers appear in the left column and the names of the members of the assembly appear in the middle column. No quantity appears since the attribute **Duplicates** is being used and the quantity is always "1."

Table Added to Report, and Then Assembly Model

Index	Component Name	Quantity
1	LARGE_PIPE	
2	SMALL_PIPE_ASSY	
3	SUPPORT_BASE	

About Obtaining a Summation

Using the **Summation** command in the TBL REGIONS menu, you can add a summation parameter to the drawing and include it in a note, or delete the summation parameter from the drawing.

To Obtain a Summation of the Parameter Values of a Repeat Region

1. Choose TABLE > **Repeat Region** > **Summation**.
2. Select a region in the drawing; then choose TBL SUM > **Add**.
3. Select a report parameter in the repeat region.
4. Type a parameter name. You must select a table cell in which to place the parameter. The system adds a new parameter to the drawing whose value is the summation of the values of the selected report parameter.
   Note: The parameter is like any other parameter that is accessible using **Parameters** in the DWG SET UP menu, except that you cannot modify or delete it.
5. When the system prompts you to update the value, choose TBL REGIONS > **Update Tables**. The system updates the table and the makes the summation parameter visible.
   Note: To update the summation parameter, you must use **Update Tables**.

Example: Summation Parameters

To create the following example, create a table for the exploded assembly, and obtain a summation of the parameter symbol value "asm.mbr.cost" in the fourth column.

At the system prompt, type the summation parameter name as cost_sum. From the keyboard, type &cost_sum in the second cell in the last row, and type Total Cost in the first cell in the last row. After you update the table, the summation appears as 60.000. To create the "total cost/part" column, add the repeat region relation total_cost_per_part = rpt_qty * asm_mbr_cost.

Note: For quantities greater than one for any particular part, you must have a "cost/part" column and a "total cost/part" column, as shown in this example.

Example of Obtaining a Summation
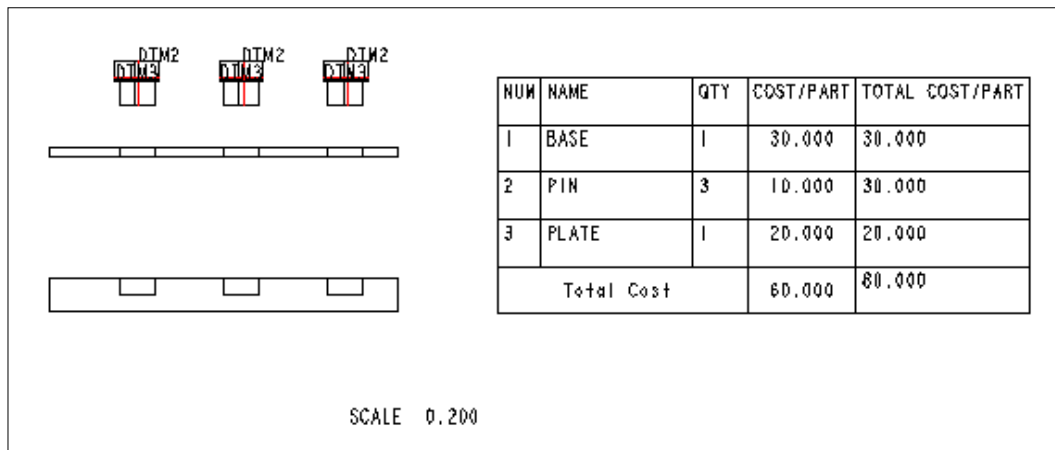
Summation Parameter Notes

You can show a summation parameter in notes just like any other drawing parameter.

However, when adding or deleting summations of parameter values, the following rules apply:

- If you delete a repeat region or table, the system deletes its summation parameter automatically.
- You cannot delete or modify summation parameters by choosing **Parameters** from the DWG SET UP menu.
- If you cannot add the report parameter values (for example, if the value types are nonnumerical), the system assigns the drawing parameter a value of "***" and displays it in that form.
- You cannot copy the summation parameter by using the **Copy Table** command in the TABLE COPY menu or by storing the table to disk.

Tip: Storing Relations and Summations with Drawing Tables

Using the **Save/Retrieve** command in the TABLE menu, you can store existing repeat region relations and summation parameters with a report table. When you retrieve a saved table containing a summation parameter into a drawing, the system checks whether the drawing already has a parameter of that name. If it cannot find one, it adds one for you. However, if it does find a parameter by that name, you must enter a name to replace it in the table. It then uses that name for the summation parameter. If you do not enter a name, it retrieves the table, but ignores the parameter.

About Nesting Repeat Regions

Nested repeat regions are repeat regions entirely contained by another repeat region. Repeat regions cannot overlap. When you nest them, you can define a double loop, such as one that searches through all subassemblies of an assembly and through all parts in each subassembly.

To Create Nested Repeat Regions

To create nested repeat regions, you define an initial *outer* region by selecting the cells to define the boundary, and adding another *inner* repeat region by selecting cells that are completely enclosed by the outer region.

You can remove an outer (enclosing) repeat region without removing the enclosed repeat region.

Examples: Nested Repeat Regions

these two cells define an outer repeat region

these two cells define an inner repeat region (note the two cells are BOTH enclosed by the outer region)
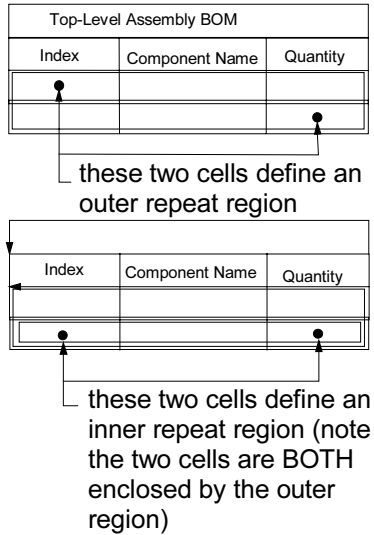
The following figure shows a table with a nested repeat region. The regions do not overlap. The attributes for both the inner and the outer repeat regions have been set using **No Duplicates** and **Flat**.

Table with a Nested Repeat Region

outer repeat region

ITEM NAME	SUB-ITEM NAME	QTY
&asm.mbr.name		&rpt.qty
	&asm.mbr.name	&rpt.qty

inner repeat region

When the model assembly "Main Vessel" is added to the report, the resulting table appears as shown in the next figure.

Model Assembly "Main Vessel" Added

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
	MAIN_PIPE	1
SMALL_PIPE_ASSY		1
	BOLT	12
	LOCK_SCREW	1
	NUT	12
	O_RING	2

	PLATE	1
	SMALL_PLATE	1
SUPPORT_BAS E		1

Entries in the outer repeat region are listed under the header "item name," while entries in the inner repeat region are listed under the header "sub-item name." Since item "Support Base" is a part and has no subitems, the row beneath its entry is blank. If you set the minimum number of repeats for the inner repeat region to 0, this prevents blank lines if there is no information at that level.

In the following figure, the attributes of the inner repeat region have been changed to **Duplicates** and **Flat**. Notice that the subitems have multiple entries where possible, and no value for quantity is listed for them.

Attributes of Inner Repeat Region Changed to Duplicates and Flat

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	MAIN_PIPE	
	LARGE_O_RING	
	LARGE_PLATE	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
SMALL_PIPE_ASSY	SMALL_PIPE	1
	PLATE	
	O_RING	
	LOCK_SCREW	
	BOLT	

	BOLT	
	BOLT	
	BOLT	
	O_RING	
SUPPORT_BASE		1

Note: Table is broken for display purposes so
as not to show all patterned bolts and nuts.

In the next figure, the attributes for the outer repeat region have been changed to **No Duplicates** and **Recursive**. The attributes for the inner repeat region have been changed to **No Dup/Level** and **Recursive**. To display the **No Dup/Level** attribute, the part "O Ring" was added to the assembly "Large Pipe" for this example only. The "O Ring" part now exists in two levels of the "Main Vessel" assembly ("Small Pipe" assembly and "Large Pipe" assembly).

Attributes of Outer Repeat Region Changed to No Dup/Level and Recursive

MAIN_VESSEL		1
	MAIN_VESSEL	1
	LARGE_PIPE	1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
	O_RING	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
NUT		12
	NUT	1

O Ring is listed twice in the table because it exists at two levels in the "Main Vessel" assembly, but is only listed once per level.

Table is broken so as to show only the area of No Dup/Level.

About Creating Two-Directional Repeat Regions

Two-directional (2-D) repeat regions are used to display information in a model's family table. These two-directional repeat regions allow tables to expand both vertically and horizontally with model data and actually consist of three separate repeat regions.

- The first of these three regions is the outermost region, in which the other two subregions are completely enclosed.
- The second and third repeat regions are one-directional regions that define how the overall two-directional repeat region expands. They define the horizontal and vertical expansion characteristics. One cell must be common to all three of these regions. This cell defines the corner boundary cell between the horizontal and vertical expansion, and it should be at the end of the two inner, subregions.

To Create a Two-Directional Repeat Region

1. Choose TABLE > **Repeat Region** > **Add** > **Two-D**.
2. Select two points to define the outer repeat region.
3. Select the cell to be the upper border of the row and column subregions.

The first cell selected is the one that remains fixed after the table expands horizontally and vertically. Next, double-click the cell that is diagonally across from the first selected cell. (For a table created as the **Descending/Rightward** type, first select the upper left cell and then double-click the lower right cell).

Tip: Deleting as a Unit

You can delete as a unit the three repeat regions that compose the 2-D repeat region.

The column and row subregions can have their own independent attributes, filters, and sorting keys.

Note: You can paginate 2-D repeat regions if they run off the bottom edge of a sheet, but not if they run off a side edge.

Examples: Two-Directional Repeat Regions
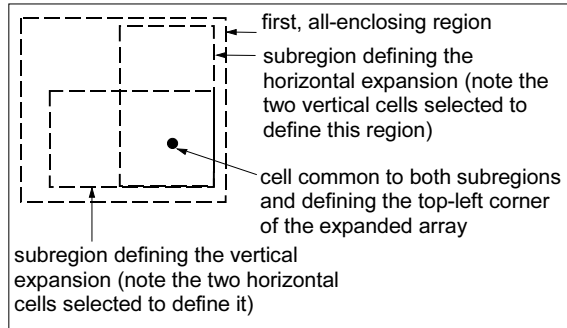
The next figure shows a table that was created to display parameter values for every instance in a family table library part.

Note: The arrangement of the repeat regions is the same as in the previous illustration. Some cells were merged to clarify the expanded table.

Table with Parameter Values for Instances in a Family Table Library Part
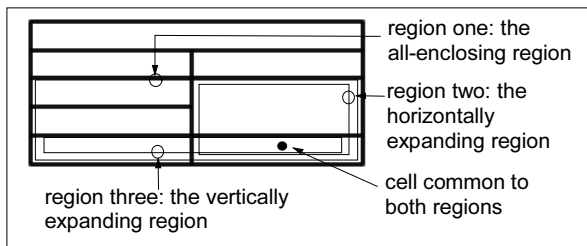
In the following figure, the heading text and parameter values were entered for the library model "ELMFEMWS.prt" .

Heading Text and Parameter Values Entered for Library Model "ELMFEMWS.prt"

This 2-D repeat region expands downward and to the right because the table was created as the **Descending/Rightward** (default) type in the TABLE CREATE menu. You can create a 2-D repeat region to expand up or down, left or right by creating the table accordingly. When part "ELMFEMWS.prt" is added to the report from the Pro/ENGINEER BASIC LIBRARY, the result appears as shown in the next figure.

You can merge the blank cells to form a continuous blank space.

The columns of the horizontally expanding region will have the same width.

To remove unwanted columns, create a filter using the syntax &fam.inst.param.name = "parameter name". For example, if the NUM_OF_TEETH parameter is not required in the table shown in the next figure, use the filter &fam.inst.param.name = "NUM_OF_TEETH".

Generic Part Name: ELMFEMHS						
	Model Parameters					
Instance Name	cutter_diam	cut_length	FLUTE_NUMBER	length	NUM_OF_TEETH	shank_dia
ELEM1	0.250	1.750	4.000	3.582	4.000	0.375
ELEM2	0.312	2.000	4.000	3.750	4.000	0.375
ELEM3	0.375	2.500	4.000	4.250	4.000	0.375
ELEM4	0.500	3.000	4.000	5.000	4.000	0.500
ELEM5	0.625	4.000	4.000	6.125	4.000	0.625
ELEM6	0.750	4.000	4.000	6.250	4.000	0.750
ELEM7	0.875	5.000	4.000	7.250	4.000	0.875
ELEM8	1.000	6.000	4.000	8.500	4.000	1.000
ELEM9	1.250	8.000	6.000	8.500	6.000	1.250
ELEM10	1.500	8.000	6.000	10.500	6.000	1.250
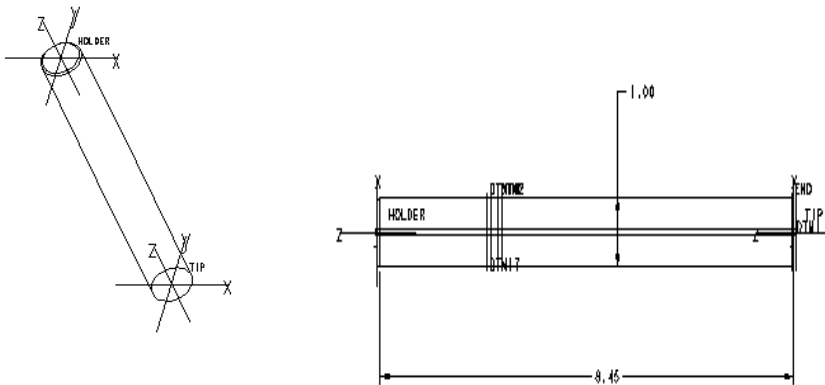
About Paginating Report Tables

You can paginate report tables that are on the same drawing sheet, and you can break a table into arbitrarily placed segments on the same sheet. As you add more information to the table, the system flows the information into each segment and adds more sheets as necessary, copying the table format (if there is one). It chooses the segments to create the new sheet as follows:

- The system copies the segment definitions from the format if the table was copied from a format, the table on the format was paginated on two sheets, and the drawing table currently has only one sheet.
- Otherwise, it copies segment definitions from what was previously the last sheet of the table.

To Paginate Report Tables That Are on the Same Drawing Sheet

1. Choose TABLE > **Pagination**.
2. Select a table that has at least one repeat region.
    Note: The **Save** command in the Pro/ENGINEER **File** menu is not available while you are in the TBL PAGIN menu.
3. If the table contains only one segment (on any sheet), choose TBL PAGIN > **Set Extent** and select a lower extent for that one segment. Otherwise, select a table segment for which to set the extent. Tables always *fill down* to use up as much of their extent as possible.
4. Choose **Add Segment** and add a new segment on the current sheet. Locate the origin of the new segment and then its extent.
5. To change the pagination, choose **Clear Extent** to clear all pagination information from the table. The

system removes all segments other than the first, appends their contents to the first segment, erases the table, and draws it in its unpaginated state. Repeat Steps 2 through 4.

6. Choose TBL PAGIN > **Done** to accept all modifications and add any remaining sheets, or choose TBL PAGIN > **Quit** to remove all modifications after responding to the confirmation.

About Creating Header and Footer Titles

When working with tables containing multiple segments, you can place a title at the top of each segment (as a header) or at the bottom of each segment (as a footer).

When creating titles, keep in mind the following rules:

- Two-directional repeat regions cannot have titles.
- For single-row headers and footers, you must select the row twice for the first and second rows.
- A region can have only one header and one footer. However, since a table can have more than one paginated region, a table can have more than one header and more than one footer.
- When selecting rows to be used in a title, you cannot choose a row that is part of a repeat region.
- Two titles cannot intersect each other. That is, when creating a new title, you cannot select a portion (one or two rows) of an existing title to be used in the new title. However, you can define the same rows to be both the header and the footer of the same region or to be titles of a number of regions.
- You cannot create a header or footer title in a row that has cells merged in a column.
- The system inserts a header at the top of a segment, and inserts a footer at the bottom of a segment, regardless of the direction of the table's growth (that is, ascending or descending).

Example: Using a Header Title in a Table with a Descending Repeat Region

The next figure illustrates how to use a header title in a table with a descending repeat region.

Example of Table Segments with Header Title in a Table with a Descending Repeat Region

INDEX	COMPONENT	QUANTITY
1	TESTPLATE	
2	PLUG	
3	PLUG	
4	PLUG	
5	WASHER_PART	

Header appears for both segments

INDEX	COMPONENT	QUANTITY
	WASHER_PART	
	WASHER_PART	
	SQUARE <NUT>	
	SQUARE <NUT>	
	SQUARE <NUT>	

To Add a Header or Footer Title to a Table

1. Choose TABLE > **Pagination**; then select a table that contains at least one repeat region.
2. Choose TBL PAGIN > **Add Title**; then select a repeat region in the current table.
3. From the REGION TITLE menu, choose **Header** or **Footer**.
4. Select the first row (one that is *not* in a repeat region) to be used as the header or footer title of the selected region.
5. Select the second row. The segments of the table appear with the specified header or footer title.

About Specifying Indentation

Using the **Indentation** command in the TBL REGIONS menu, you can specify an indentation for report data in recursive repeat regions. The cell to which you apply the indentation must be in a repeat region that has **Recursive** specified as one of its attributes, and should contain a recursive report parameter symbol (such as "asm.mbr.name"), although this is not a requirement. If you select a cell that does not contain a recursive parameter, the system informs you, but it indents the data if you specify a value.

Pro/ENGINEER expresses indentation for a repeat region cell using a number between 0 and 100. The number indicates the value in characters (at the default text size) by which that data is indented in a recursive repeat region. Pro/ENGINEER increments the indentation by the specified amount at each new level in the table.

To Specify Indentation for a Repeat Region Cell

1. Choose TBL REGIONS > **Indentation**.
2. Select the cell in a repeat region template for which you want to specify the indentation.
3. Type a value for the indentation.
4. Choose **Update Tables** to redisplay the table with indentation.

Note: If you specify the indentation for a cell in an existing table, you must choose TBL REGIONS > **Update Tables** before you can view the result.

Example: Repeat Region Cells Using Indents

Indentation has been specified for the second column, containing values of "asm.mbr.name," and the third column, containing values of "asm.mbr.type." For each cell, the indentation was specified as 5. Note how this indentation is incremented at each recursive level.

Indented Information in a Recursive Repeat Region

LEVEL	NAME	TYPE
1	MAIN_VESSEL	ASSEMBLY
2	LARGE_PIPE	ASSEMBLY
3	MAIN_PIPE	PART
3	LARGE_O_RING	PART
3	LARGE_PLATE	PART
3	LGE_BOLT	PART
3	LGE_BOLT	PART

Note: table broken for display purposes so as not to show all patterned bolts and nuts

3	LGE_NUT	PART
2	SUPPORT_BASE	PART
2	SMALL_PIPE_ASSY	ASSEMBLY
3	SMALL_PIPE	PART
3	PLATE	PART
3	O_RING	PART
3	LOCK_SCREW	PART
3	BOLT	PART

3	BOLT	PART
3	BOLT	PART
3	O_RING	PART

About Adding Filters

Using the **By Rule** command in the FILTER REG menu, you can remove multiple items that match a specified pattern. You can use filters in the following forms to further specify the information to appear:

- *<symbol>*—Any parameter that is valid in a repeat region

- *<comparison operator>*—Any of the operators <, >, <=, =>, ==, and !=

- *<literal value>*—Any integer, floating point, or string value

Filters exclude from a repeat region any record that uses one of the filtered parameters and whose value does not match the constraint defined by the filter. The system omits the entire record from the table, not just the parameter affected by the filter. For instance, the filter &asm.mbr.type==part in a repeat region omits all records of objects other than parts from the report.

Note: If you filter in a two-directional repeat region, it only removes the particular cell, not the entire record.

A filter such as &asm.mbr.material!=steel excludes all records of assembly members with a material parameter value (user-defined) of "steel" from the report.

You can also enter filters that work for multiple values, such as &asm.mbr.name==part_a,part_b,part_j, which would exclude from the report all records of assembly members other than "part_a.prt", "part_b.prt", and "part_j.prt". A line can contain up to 80 characters. When creating a filter with multiple acceptable values such as this one, you can only use the operators "==" and "!=".

When using filters with these operators in a repeat region, keep in mind the following:

- If you add filters with the operator "==" to more than one line in a repeat region, all entries are blanked and the table appears to be empty.
- For "==" operators with more than one value, the values are linked by "or." For example, for the filter &asm.mbr.name==part_1,part_2,part_3, the repeat region would exclude all objects other than those parts having the name "part_1," "part_2," or "part_3."
- For "!=" operators with more than one value, the values are linked by "and." For example, for the filter &asm.mbr.name!=part_a,part_b,part_c, the repeat region would exclude those parts having the name "part_a," "part_b," and "part_c."

About Using Wildcard and Backslash Characters in Filters

You can use wildcard characters (*) in report filters, but they are only allowed in filters that use the operators "==" or "!="; filters of any other type that contain "*" are in error, and you are prompted to re-type the filter. For example, you can use the wildcard in these filters:

- &asm.mbr.name==part*
- &asm.mbr.name==*my*, *your*

The first filter would match the strings "part," "part1," "part_A," and "partabcdefg." The second filter would match the strings "my," "this_is_my_assembly," "autonomy," "not_yours," and "your."

If you add a backslash (\) in the right-hand side of a filter (after the operator), the system reads the character after the backslash literally (as itself), not as a special character. You can then filter for an asterisk character. For example, &asm.mbr.name>=part* matches strings that are alphanumerically greater than or equal to the string "part*".

The system interprets backslashes in report parameters or between string quotes literally as backslash characters. It interprets the parameter "&asm.\mbr.name" as a report symbol named "asm.\mbr.name" (an invalid name), and the filter:

&asm.mbr.name=="match this\"

matches only the string "match this\".

It treats a backslash that is not in a report parameter, not enclosed by string quotes, and not followed by another character as a null string (" "). It interprets anything between two string quotes literally; the filter:

&asm.mbr.name<":***"

matches strings that are alphanumerically smaller than ":***". For filters created prior to Release 11.0 that contain the wildcard character, the system interprets the character literally (as an asterisk).

Note: You should not use filters on the system parameter symbol "&rpt.index".

To Add a Filter to a Repeat Region

1. Choose TBL REGIONS > **Filters**.
2. Select the repeat region to which you want to add a filter.
3. Choose FILTER REG > **Add**. Type the filter expression.
4. Choose **Done** to add the filter. Existing repeat regions regenerate to accommodate the filter.

Examples: Using No Dup/Level and Recursive Attributes

The attributes for the inner repeat region are specified as **No Dup/Level** and **Flat**.

Attributes for Outer Repeat Region Specified as No Dup/Level and Recursive

ITEM NAME	SUB-ITEM NAME	QTY
MAIN_VESSEL		1
	LARGE_PIPE	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

When the filter &rpt.level>1 is added to the outer repeat region, the resulting table looks like the example in the next figure. The row containing "Main Vessel," the first level of the assembly, is removed. Only those members with a level value higher than 1 remain.

Filter "&rpt.level>1" Added to Outer Repeat Region

ITEM NAME	SUB-ITEM NAME	QTY
MAIN_VESSEL		1
	LARGE_PIPE	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

Moving a Filter to Another Repeat Region

In the following figure, the filter asm.mbr.type==assembly is added to the inside repeat region, so no parts are listed there.

Filter "asm.mbr.type==assembly" Added to Inside Repeat Region

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

In the next figure, that filter is cleared from the inside repeat region and added to the outside repeat region. Again, all parts have been omitted from the region.

Filter Cleared from Inside Repeat Region and Added to Outside.

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
SMALL_PIPE ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12

About Excluding Items from a Repeat Region

You can exclude items from a repeat region by graphically selecting them in the report using the FILTER TYPE menu. However, when removing selected items from a repeat region, the following restrictions apply:

- The system preserves the exclusion attribute if you rename the model by choosing **Rename...** from the Pro/ENGINEER **File** menu or using Pro/PDM.
- You cannot filter items if the table is frozen.
- This functionality is not available in Pro/DIAGRAM.
- You cannot exclude the following:
  - Items in 2-D repeat regions and subregions (you can use **By Rule** to filter out columns or rows in 2-D repeat regions).
  - Records that have comment cells and dash items.
  - Records that have no associated parametric attachment information.

- The system does not preserve the exclusion attribute in the following situations:
  - When the region has the **No Duplicates** attribute and the assembly does not have at least one copy of the excluded item during the assemble/disassemble process.
  - If you switch among the **Duplicates**, **No Duplicates**, and **No Duplicates/Level** attributes.

To Remove Selected Items from a Repeat Region

1. Choose TBL REGIONS > **Filters** > **By Item** > **Exclude** > **Pick Many**.
2. Select repeat region records to exclude. Be sure that the selection box encloses the outer boundary of the region on both sides.

Example: Excluding Items from a Repeat Region

Select these records to exclude.

index	type	name	level	qty
1	PART	TESTPLATE	1	
2	PART	PLUG	1	
3	PART	PLUG	1	
4	PART	PLUG	1	
5	PART	WASHER_PART	1	
6	PART	WASHER_PART	1	
7	PART	WASHER_PART	1	
8	PART	SQUARE	1	
9	PART	SQUARE	1	
10	PART	SQUARE	1	

The system updates the repeat region to show the new order.

index	type	name	level	qty
1	PART	TESTPLATE	1	
2	PART	SQUARE	1	
3	PART	SQUARE	1	
4	PART	SQUARE	1	

About Setting Recursive or Flat Items in a Repeat Region

You can select individual items graphically in a repeat region to make them *recursive*—displaying items within the assembly—or *flat*—not displaying items within the assembly. When you set an item as flat or recursive, the system *marks* it until you choose **Default** or **Default All** to change it. Items that you set individually remember their setting even if you change the default attribute of the region from **Flat** to **Recursive** or vice versa.

To Set an Item as Flat

1. Choose TBL REGIONS > **Flat/Rec Item** > **Flat**.
2. Select a record in the table. Use **Pick Many** to select more than one. Be sure that the selection box encloses the outer boundary of the region on both sides.
3. Choose **Done Sel**. The system updates the repeat region.

Example: Setting an Item as Flat

index	name	type	level	qty
1	NEWTOP	ASSEMBLY	1	
2	P1	PART	2	
3	SUB1	ASSEMBLY	2	
4	P2	PART	3	
5	SUB1A	ASSEMBLY	3	
6	P3	PART	4	
7	P4	PART	4	
8	SUB2	ASSEMBLY	2	
9	P5	PART	3	
10	P6	PART	3	

Select these items to make flat.

Recursion stops at these items.

index	name	type	level	qty
1	NEWTOP	ASSEMBLY	1	
2	P1	PART	2	
3	SUB1	ASSEMBLY	2	
4	SUB2	ASSEMBLY	2	

Example: Setting an Item as Recursive

Each record is recursed.

Select these two records
to make recursive.

index	name	type	level	qty
1	P1	PART	2	
2	SUB1	ASSEMBLY	2	
3	SUB2	ASSEMBLY	2	

index	name	type	level	qty
1	P1	PART	2	
2	SUB1	ASSEMBLY	2	
3	P2	PART	3	
4	SUB1A	ASSEMBLY	3	
5	SUB2	ASSEMBLY	2	
6	P5	PART	3	
7	P6	PART	3	
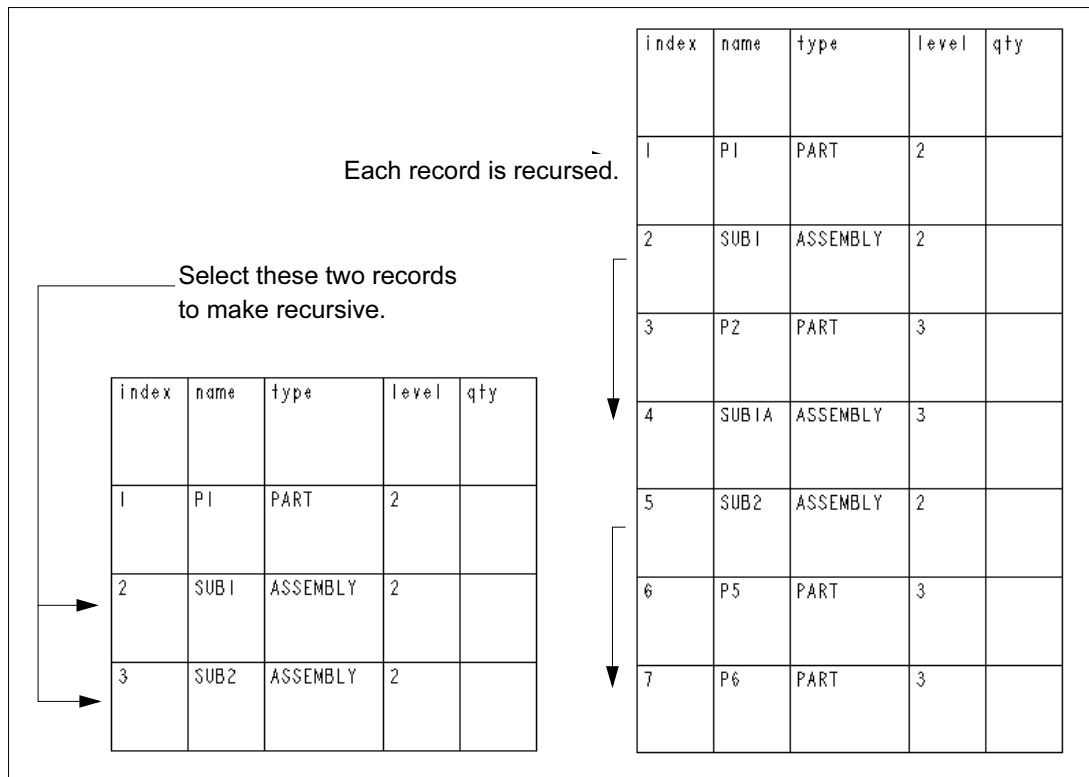
To Set an Item as Recursive

1. Choose TBL REGIONS > **Flat/Rec Item** > **Recursive**.
2. Select a record in the table. Use **Pick Many** to select more than one record. Be sure that the selection box encloses the outer boundary of the region on both sides. You cannot select the following:
- Items in 2-D repeat regions or subregions.
- Records that cannot have comment cells and dash items.
- Records that do not have any associated parameter attachment information.
3. Choose **Done Sel**. The system updates the repeat region to display the new order. The recursion starts one level from the item that you select.

About Sorting in a Repeat Region

Using the SORT REGION menu, you can sort the contents of a repeat region to change the order in which the system lists entries in a table:

- When you choose **Default Sort**, it sorts entries forward, by ASCII character value.
- When you choose **No Default**, it sorts entries in database order.

You can specify more than one parameter symbol for sorting a region; the system sorts entries by the first parameter, and then by each succeeding parameter, if necessary. You can enter user-defined parameters as sorting parameters. You can sort a region forward (in ascending order) or backward (in descending order).

For the purposes of sorting, the system considers a text string in a repeat region to consist of two parts:

- A nonnumerical part, which comprises everything in the string that precedes the numerical part.
- A numerical part, which comprises all contiguous numbers at the end of the string, possibly including a decimal point.

> **Note:** The system considers a leading "+" or "-" to be a portion of the numerical part of the string if the string contains no other nonnumerical data.

In any given string, either the numerical or the nonnumerical part can be empty. The system determines the sort order by making a comparison between each pair of strings in the field selected as the sort key. Instead of comparing each pair of strings strictly as strings, it considers the numerical and nonnumerical parts separately. If the nonnumerical parts of two strings are identical, it determines the order by considering their numerical parts as numbers. A sorting parameter is valid only for the specified repeat region. Repeat regions that are nested are not affected by sorting parameters at a higher region.

To Add a Sorting Parameter to a Region

1. Choose TBL REGIONS > **Sort Regions**.
2. Select within the region you need to sort.
3. Choose SORT REGION > **Add**.
4. From the SORT ORDER menu, choose **Forward** or **Backward**.
5. Select the repeat region parameter by which you want to sort. Choose **Done Sel** when you have finished.
6. Choose SORT REGION > **Done**. The system reorders the entries in the repeat region.

About Sequentially Indexing Separate Repeat Regions

You can link the indexing of separate repeat regions in the same table by using the REGION ATTR menu. By doing so, you can specify that the index numbering for one repeat region should continue in sequence from where the numbering for another repeat region ended.

To Link the Indexing of Two Repeat Regions

1. Choose TBL REGIONS > **Attributes**.
2. Select inside the repeat region for which you want to set the indexing.
3. Choose REGION ATTR > **Start Index**. Select the repeat region from which you want to continue the indexing.
4. Choose **Done/Return** to finish; the system updates the value of "&rpt.index" for the selected repeat region.

Note: You cannot use sequential indexing within nested repeat regions.

Example: Sequentially Indexing a Report with Two Repeat Regions

In the following report, the index number in the second region begins at 1. This might not be desirable, since the assemblies listed in the second region are still members of assembly "Main Vessel," like the parts in the first region. The **Start Index** command continues the numbering in the second region in sequence from where the numbering in the first region ends.

Index Number in the Second Region Beginning at 1

INDEX	NAME	TYPE	QTY
1	SUPPORT_BASE		1
INDEX	NAME	TYPE	QTY
1		LARGE_PIPE	1
2		SMALL_PIPE_ASSY	1

To Display the Table Shown Next

1. Choose TBL REGIONS > **Attributes** and select the second repeat region.
2. Choose **Start Index** and select the first repeat region.
3. Choose **Done/Return**. The system updates the index numbering of the second repeat region so that it continues the numbering begun in the first region. To return the index numbering to its original state so that it starts at 1, choose REGION ATTR > **No Start Idx**.

Example: Updated Index Numbering of Second Repeat Region

INDEX	NAME	TYPE	QTY
1	SUPPORT_BASE		1
INDEX	NAME	TYPE	QTY
2		LARGE_PIPE	1
3		SMALL_PIPE_ASSY	1

About Fixing an Index

Using the **Fix Index** command in the TBL REGIONS menu, you can *fix* the index of a repeat region record so that it remains the same even after you insert additional items into the repeat region or sort the repeat region differently. When fixing an index of a repeat region, keep the following restrictions in mind:

- If you fix the index of a record to be larger than the size of the repeat region, that record appears at the end of the repeat region.
- If you remove a record from a repeat region whose index has been fixed (for example, a component is suppressed in an assembly), the fixed index does not appear in the repeat region until you unfix it or use it for another record.
- If you change the attribute of a repeat region from **Duplicate** to **No Duplicate** (or vice versa), the fixed index no longer appears. However, if you change the attribute back to its original setting, the system replaces the fixed index.
- You cannot use an index with these symbols:
  - All symbols of the type "asm.mbr.cparam"
  - All symbols of the type "asm.mbr.cparams"
  - All symbols of the type "asm.mbr.cblprm"
  - All symbols of the type "asm.mbr.cblprms"
  - All harness symbols
  - All family table symbols
  - All symbols showing cabling terminator names or types
- If a record cannot have comment cells and dash items, you cannot fix its index.
- If you fix a record's index, you cannot dash its "rpt.index," and vice versa.
- You cannot use a fixed index for 2-D repeat regions.
- You cannot fix the index of a process symbol (that is, all symbols of type "prs").
- A fixed index takes precedence over the following:
  - Report relations.
  - Start index of a repeat region. For example, if a repeat region starts at index 12 (taken from the last index of another repeat region), but one of its records is fixed at 2, that record appears first in the repeat region with index 2.
  - Sort keys of a repeat region. The system determines the position of a record by its fixed index if it has one. For example, a record is always at the beginning of a repeat region if its index is fixed to 1.

To Fix the Index of a Repeat Region

1. Choose TBL REGIONS > **Fix Index**; then select a repeat region. The FIX INDEX menu displays the following commands:
*   **Fix**—Fixes one or more indexes for one or all records in a repeat region.
*   **Unfix**—Unfixes an index, all indexes in a record, or all indexes in a repeat region.
*   **Record**—Fixes or unfixes one or all indexes in a record of a repeat region (use with **Fix** or **Unfix**).
*   **Index**—Unfixes an index (this command is only available if you choose **Unfix**).
*   **Region**—Fixes all indexes in all records in a repeat region, or unfixes all fixed indexes in a repeat region (use with **Fix** or **Unfix**).
2. Choose **Fix** and **Record**. Select a record in the current repeat region (to fix the indexes in all records in a repeat region, choose **Fix** and **Region)**.
3. Type the desired index for that record (you cannot use indexes that are already fixed for other records).
4. Choose **Done**. The system fixes the index for that record.
    Note: To display any changes that you make using the commands in the FIX INDEX menu, you must choose **Done**.

To Unfix an Index of a Repeat Region

*   To unfix all indexes in a record, choose **Unfix** and **Record**. The system highlights all records whose indexes are fixed. You can then select the records that you want to unfix.
*   To unfix one index, choose **Unfix** and **Index**; then type a fixed index.
*   To unfix all fixed indexes in all records in a repeat region, choose **Unfix** and **Region**; then choose **Confirm** from the CONFIRMATION menu.

About Adding Comment Cells

A comment cell is a cell in a repeat region that contains user-supplied text rather than data that is read from a model. Using comment cells, you can annotate data in a row of a repeat region and your additional text remains associated with that row, even if the row's location within the region changes.

Pro/ENGINEER tracks a comment cell to a particular model (not a parameter value), so that when the model name changes, the comment is not lost. You can use comment cells in all reports except those for family tables and cable harnesses. However, you cannot use comment cells with the following parameter symbols:

*   All parameter symbols of the following type:
    – "asm.mbr.cparam"
    – "asm.mbr.cparams"
    – "asm.mbr.cblprm"
    – "asm.mbr.cblprms"
    – "PRS"
*   All harness parameter symbols
*   All family table symbols
*   All parameter symbols showing cabling terminator names or types

Before creating a comment cell, you must list the data in the table so that you can choose a particular object to comment.

To Create a Comment Cell

1. Choose TBL REGIONS > **Comments** > **Define Cell**.
2. Select an empty cell within an existing repeat region template. In Report and Format mode, the system highlights the cell in cyan. All cells in that column are now comment cells.

3. Choose TABLE > **Enter Text** and type text in the comment cell for a particular entry. The text remains associated with that entry regardless of how the system sorts the region.

To Delete a Comment Cell

1. Choose RPT COMMENTS > **Clear Def**.
2. In the repeat region template, select a comment cell to delete. The system removes all comment text from this cell and from all of the comment cells that follow it.

Whenever you remove a row in the repeat region from the table (either by using a filter or by changing the setting for **Duplicates/No Duplicates/No Dup/Level** or **Recursive/Flat**), the system deletes the text of a comment cell in that row. It does not redisplay the text if you return the row to the table by clearing the filter or resetting the region attributes.

To Use Dash Items

Using the **Dash Item** command in the TBL REGIONS menu, you can convert selected "rpt.qty" and "rpt.index" values in a drawing or report table to a dash "-". To remove a dash symbol from the table, select the symbol and the appropriate parameter value redisplays in the table.

You cannot use a Dash item with the following parameter symbols:

- All parameter symbols of the following type:
  - "asm.mbr.cparam"
  - "asm.mbr.cparams"
  - "asm.mbr.cblprm"
  - "asm.mbr.cblprms"
- All harness parameter symbols
- All family table symbols
- All parameter symbols showing cabling terminator names or types

Whenever you change a table value to or from a dash symbol, the system updates the table. If "rpt.qty" or "rpt.index" parameter symbol values appear more than once in a repeat region, and you convert any of them to a dash symbol, the system converts all like occurrences of these values in that region to a dash symbol. Likewise, if you remove a dash symbol from a table region containing multiple occurrences of the same parameter value, the system converts all like parameter values containing dashes.

When a "rpt.index" parameter symbol value converts to a dash symbol, it does not disturb the continuity of the remaining index values. For example, if the system converts the third record in a table index to a dash symbol, the index appears as 1, 2, -, 3, 4, ... If it converts any "rpt.qty" parameter symbol value in a table to a dash symbol, it has no effect on the table's evaluation of relations, sorting, or filtering. The table evaluates relations, sorts, and filters as if the true quantity value was displayed. BOM balloons containing "rpt.qty" or "rpt.index" parameter symbol values that have been replaced in the table by a dash symbol do not appear until you remove the Dash symbol in the table.

Tip: Dash Symbol Associativity

Dash symbols are associated with the data they replace, just as table comments are, and can be lost if you modify the repeat region's attributes. For example, if you replace a report quantity value of 3 with a dash symbol, and then modify the region's attributes to duplicates, the system warns you that the content of comment cells or dash items can be lost and asks you if you want to continue. If you continue, it converts the dashed symbols to the appropriate parameter value display.

About Writing Relations

Using the **Relations** command in the TBL REGIONS menu, you can write relations among parameter symbols in a repeat region and output the computed information in the same repeat region. The system stores the relations' parameter symbols created in a repeat region with it and you cannot reference them outside of the region. In assignment statements, you can put only new parameter symbols on the left-hand side. You must refer to parameter symbols in repeat regions by specifying their full name and converting the period (.) to an underscore (_), as shown in the following figure. To use the following example, you *must* add "rpt_qty" as a repeat region parameter symbol.

Example of Relation

```
IF asm_mbr_type=="BULK ITEM"
    Qty=50
ELSE
    Qty=rpt_qty
ENDIF
```

To Write a Relation Among Parameter Symbols in a Repeat Region

1. Choose TBL REGIONS > **Relations** > **Add**.
2. Create a relation using the report parameters of the selected region, or create new parameters to show in the same region.
   Note: If a report parameter is not in a repeat region, you can use it to create a relation if you add it to the relation set first by choosing RELATIONS > **Add Param**.
3. Choose **rpt**, **rel**, and **User Defined** from the REPORT SYM menu to use this relation in the repeat region.
4. Type the name of the parameter in the repeat region relation.

Tip: Accessing Dimension Values

To access dimension values in a repeat region, use the report parameter symbol "asm.mbr.d*". To use them in a relation, type param1 = asm_mbr_d1 * asm_mbr_d2. If the report parameter symbols "asm_mbr_d1" and "asm_mbr_d2" are not in the repeat region text, you can add them as parameters into the repeat region relation first by choosing **Add Param** from the RELATIONS menu. You can only access dimension values of the *current* model.

About BOM Balloons

Using the BOM BALLOON menu, you can display BOM balloons for any model in a drawing or report that has a repeat region associated with it. You can show balloon notes for members of an assembly in a view so that they visibly correspond to a record in a repeat region containing BOM information. Each balloon contains a number that corresponds to that member's index number (the value of &rpt.index) in the repeat region (by default), and can also contain a quantity for that member as well (the value of &rpt.qty).

Pro/ENGINEER updates the contents of the balloon note to reflect changes in the order or quantity of the corresponding records. It also updates BOM balloons to reflect the addition or removal of assembly components. When you are working with BOM balloon notes, the following restrictions apply:

- BOM balloons are available only for non-nested repeat regions.
- Erasing or deleting balloon notes removes them from the display, but you can redisplay them by showing them.
- BOM balloons do not display on another view if you erase the view to which they are attached.

- You cannot modify the size of a quantity BOM balloon. However, you can modify a simple BOM balloon size by changing the drawing setup file options `min_balloon_radius` and `max_balloon_radius`.
- You can show BOM balloons only for model components in an assembly. The system does not consider the spools and tie wraps used in cabling assemblies to be assembly model components, and you cannot assign BOM balloons to them even though you can display them in a table with a report index value. It does consider cabling harnesses to be assembly model components, and you can assign BOM balloons to them.
- BOM balloons are supported only when the model name is listed in the repeat region of a table.

Regenerating Balloons for Suppressed Components

The system preserves report cosmetics such as BOM balloons, dash items, and comments of suppressed assembly members. When you resume suppressed features, it also restores the report cosmetics on the drawing associated with them. For BOM balloons of suppressed components, Pro/ENGINEER attempts to find a placement of the same model that is not suppressed and reroute the balloon to that placement. If it is unable to find the placement, it erases the balloon, but it resumes when you resume the component.

When you suppress or replace a component to which a BOM balloon is attached, **Bln By Part** reattaches the balloon to another placement of the same part.

Bln By Comp specifies that simple BOM balloons reattach themselves to whatever component replaced the one that originally owned the BOM balloon.

When you set a region using the **Bln By Comp** command in the TBL REGIONS menu, the system regenerates simple BOM balloons as follows:

- If the component that owned the BOM balloon no longer exists, the system deletes the balloon.
- If you suppress the component, it suppresses the balloon.
- If you replace the component with another component, and there is not already a balloon for the model that replaced the original component, the balloon becomes a balloon for the replacement component.
- The balloon's position on the drawing is not affected, but the leader attachment does change.
- If the replacement component already has a balloon, the system suppresses the balloon for the original component. It resumes it if some other component without a balloon replaces the replacement component.

Balloons can properly forward their attachments in this manner for automatic family table replacement. For the following situations, the system deletes the balloon for the replaced component (but you can show a new balloon in the usual manner):

- Manual replacement
- Replacement by interchange
- Substitutions by simplified representations other than with a simplified representation of the same part
  Note: These BOM balloon rules do not apply to quantity or custom BOM balloons.

To Change BOM Balloon Index Numbers

The order of the records in the repeat region controls the index numbers for BOM balloons. You can change these numbers by entering or changing the region's sorting parameters or by fixing the indexes of report items.

You can also specify a different parameter for the index field by choosing **Set Param** from the BOM BALLOON menu.

Examples: Changing the Display of BOM Balloons

The next figure shows the default display of simple (index-displaying) BOM balloons for a drawing with two assemblies; each repeat region is sorted in the default manner, in alphabetical order.

ASSEMBLY: ASM_1				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BASE	PART	1	4.500
2	PIN	PART	6	0.500
3	PLATE	PART	1	1.750

ASSEMBLY: ASM_2				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BOTTOM	PART	1	3.500
2	COVER	PART	1	1.350
3	INSERT	PART	4	0.650
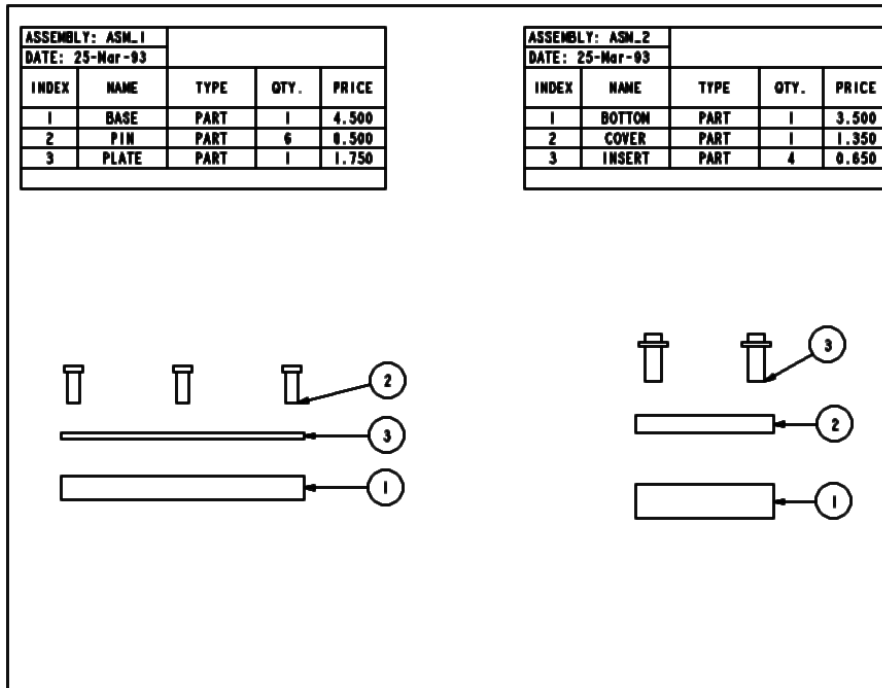
The next figure shows how the system updates BOM balloons for ASM_1 when using the sorting parameter symbol &asm.mbr.price to sort its repeat region instead (price is a user-defined parameter).

ASSEMBLY: ASM_1				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	PIN	PART	6	0.500
2	PLATE	PART	1	1.750
3	BASE	PART	1	4.500

ASSEMBLY: ASM_2				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BOTTOM	PART	1	3.500
2	COVER	PART	1	1.350
3	INSERT	PART	4	0.650
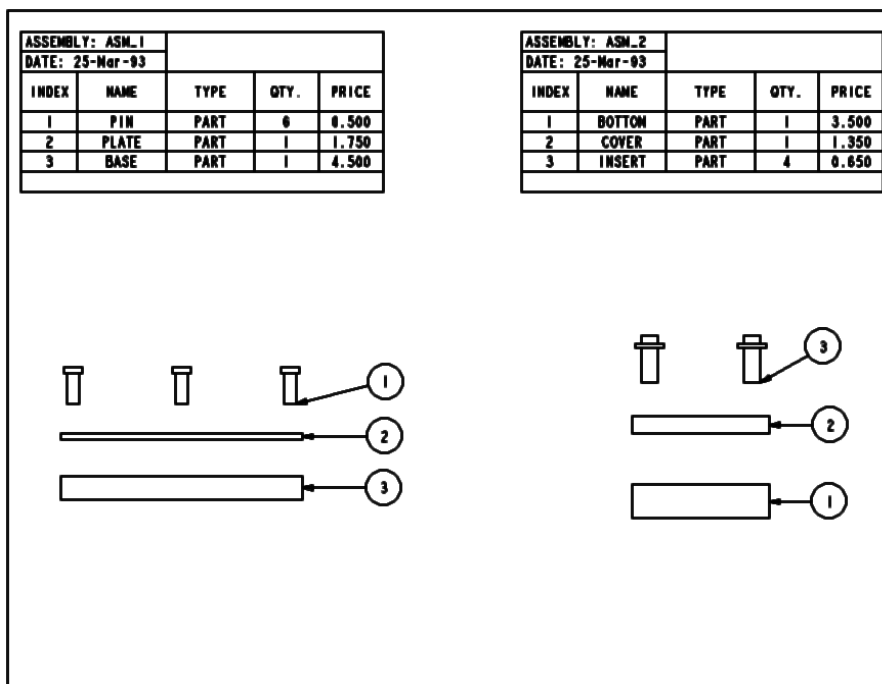
Quantity BOM balloons display both the index of the assembly member and its quantity in the assembly. The first figure that follows shows the display of quantity BOM balloons for ASM_1. Notice that the index number is in the top half of the balloon and the quantity is in the bottom half.

Note: You cannot mix simple BOM balloons and quantity BOM balloons for the same repeat region; the balloons must all be of a single type.

ASSEMBLY: ASM_1				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BASE	PART	1	4.500
2	PIN	PART	6	0.500
3	PLATE	PART	1	1.750

ASSEMBLY: ASM_2				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BOTTOM	PART	1	3.500
2	COVER	PART	1	1.350
3	INSERT	PART	4	0.650
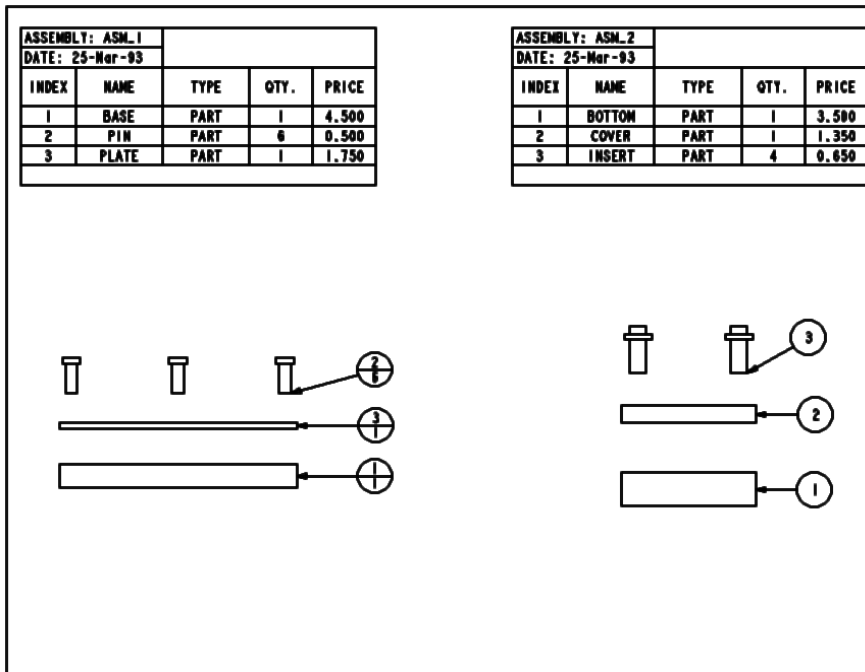
In the second figure, the quantity for the pin part is redistributed among three balloons, each pointing to two occurrences of the model.

ASSEMBLY: ASM_1				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BASE	PART	1	4.500
2	PIN	PART	6	0.500
3	PLATE	PART	1	1.750

ASSEMBLY: ASM_2				
DATE: 25-Mar-93				
INDEX	NAME	TYPE	QTY.	PRICE
1	BOTTOM	PART	1	3.500
2	COVER	PART	1	1.350
3	INSERT	PART	4	0.650
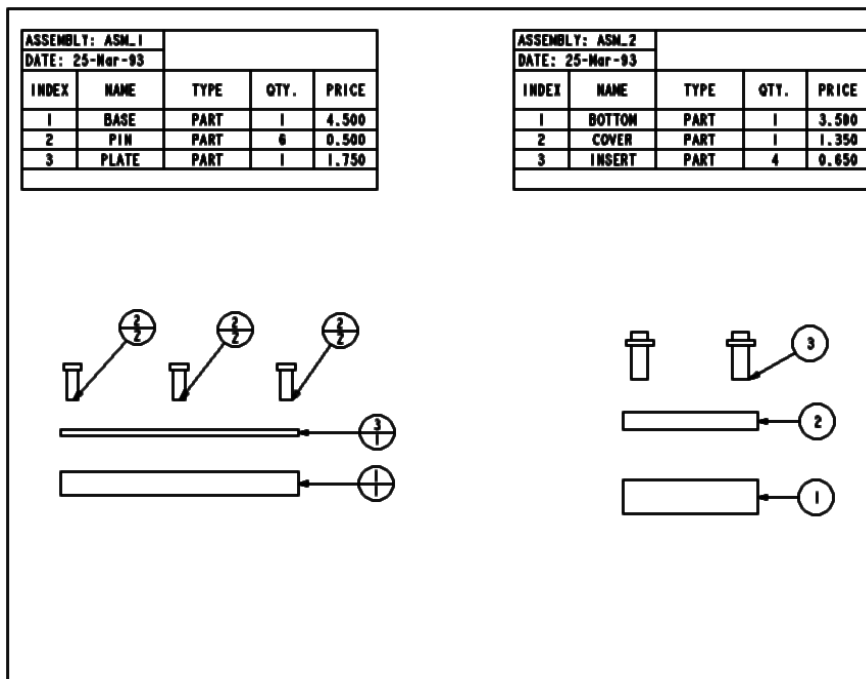
Showing BOM Balloons

You can show BOM balloons for any model in a drawing or report that is associated with a repeat region, as well as show BOM balloons in assembly simplified representations using the same method.

You can also show BOM balloons for multiple repeat regions in a single table for the same model or in multiple tables for multiple views of the model. Also, by creating multiple repeat regions, using filters, and fixing

indexes, you can show different parameters for a model in a table, but show the same BOM balloon callouts on a view.

When showing BOM balloons, keep in mind the following:

- You can show BOM balloons only on views to which the repeat region is associated. You cannot switch BOM balloons shown on one simplified representation to a different simplified representation.
- After a simplified representation is attached to a repeat region, if you change the simplified representation (that is, substitute a part with a simplified representation), all references remain with it, including the following:
  - Balloon attachment
  - Parameters
  - Comments
  - Fixed indexes
  - Dash items

To Show BOM Balloons

1. Retrieve a report, drawing, or layout that already contains a table with a repeat region.
2. Choose TABLE > **Bom Balloon > Set Region > Show > By Comp**.
3. Select a component in the drawing (for example, a part plate); then choose **Done**. The BOM balloon appears in the drawing.

Example: Showing the Part Plate Balloon

Note: The BOM balloon (number 2) attached to the part component.
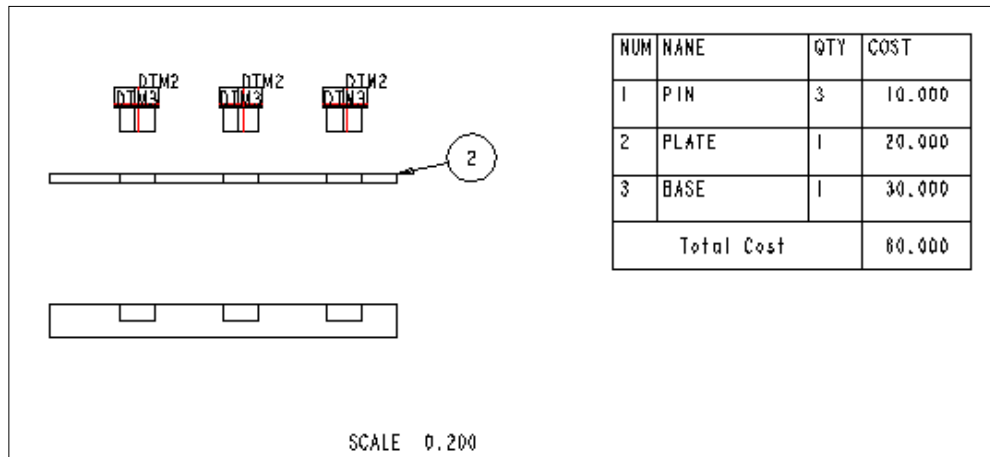
Tip: Setting the Default Arrow Style for BOM Balloons

Note: To set the default arrow style for BOM balloons in reports, set the drawing setup file option "def_bom_balloon_leader_sym."

To Change the Type of BOM Balloon Displayed

1. Choose TABLE > **BOM Balloon** > **Change Type**.
2. Select the repeat region for which you want to change the BOM balloon type. The system highlights the region and the BOM BAL TYPE menu appears with the current type highlighted.
3. Choose **Simple** or **With Qty**, followed by **Done/Return**.

4. Choose BOM BALLOON > **Show** to redisplay the BOM balloons in their new type. If multiple models are present in the drawing or report, you must select the repeat region again for the new balloons to appear.

To Change the Default Balloon Symbol for a Repeat Region

To change the default balloon symbol for a repeat region, choose Change Type from the BOM BALLOON menu and Custom from the BOM BAL TYPE menu; then select a new symbol.

Creating Customized BOM Balloons

Using the **Custom** command in the BOM BAL TYPE menu, you can create customized balloons with user-defined symbol instances. When you use a customized symbol to show balloons, the system substitutes variable text with corresponding report parameters. If the default value of the variable text matches a report symbol specification, it displays the value of that report symbol in the symbol.

For example, if the variable text default value is asm.mbr.name, the system displays the member name in the balloon. It updates (in the symbol instance) only the report parameters that it displays in the balloon region; other variable texts simply display their default values. In addition to report parameters displayed in the table, the system uses two special keywords as default values for variable text: it replaces "qty" with the quantity, as in default quantity balloons, and it treats "index" as the index field in default balloons. By default, for variable text whose default value is "index," it fills them in with the value of "rpt.index."

You can use the **Set Param** command in the BOM BALLOON menu to specify a different parameter for the index field. The system considers a customized balloon to be a quantity custom balloon if it contains variable text whose default value is "qty."

To Create a Customized Balloon

1. Choose REPORT > **Table** > **BOM Balloon** > **Change Type**.
2. Select the region where you want to change the balloon.
3. Choose BOM BAL TYPE > **Custom** > **Retrieve** and retrieve user symbols from a file.
   Note: You cannot select user-defined symbols that do not contain variable text.

To Replace An Individual Custom Symbol with Another Custom Symbol

1. Choose BOM BALLOON > **Alt Symbol**; then select a balloon region.
2. Select quantity or custom balloons of the selected repeat region, or components; then choose GET SELECT > **Done Sel**.
3. From the GET SYMBOL menu, select a user-defined symbol. Any selected balloons change to match the selected symbol, and the system redisplays the new balloons using the selected symbol for any selected components.
4. For regions that use a user-defined symbol to display balloons, choose GET SYMBOL > **Reg Default**. The selected balloons revert to the default symbol of the region, and the system shows new balloons for the selected components using that symbol.
   Note: You cannot change individual simple-customized balloons to quantity-customized balloons, and vice versa. All balloons for any given region must be either quantity or simple balloons.

Splitting and Merging Balloons

To split and merge customized balloons, use the same method you would use to split and merge quantity balloons, except that you need not enter quantity-related information. When you first merge custom or quantity balloons, the system orients them horizontally relative to the balloon to which they are merged. You can change the orientation of the balloon with respect to its parent by using the **Move** command in the DETAIL menu.

To Merge Quantity or Custom BOM Balloons

1. Choose BOM BALLOON > **Merge**.
2. Select a *from* balloon and a *to* balloon. They need not point to the same model. The *from* balloon can be a *to* balloon from a previous merge.

- If the *from* and *to* balloons point to the same model, the system deletes the *from* balloon and adds its quantity to the *to* balloon.

- If the balloons point to different models, it deletes the leader of the *from* balloon and attaches the *from* balloon to the *to* balloon.

- If you later use **Detach** to separate the merged balloons, the *from* balloon reattaches itself to its original model.

    Note: To change the orientation of a merged BOM balloon with respect to its parent, choose DETAIL > **Move**.

To Redistribute Quantity Among BOM Balloons

1. Choose BOM BALLOON > **Redistribute**.
2. Select a BOM balloon with a quantity greater than 1.
3. Type the quantity that you want to subtract from this balloon and add to another balloon for the same model.
4. Select the balloon for the same model to which you want to add the quantity. The quantities of the balloons change accordingly.

To Split Quantity BOM Balloons

1. Choose BOM BALLOON > **Split**.
2. Select a quantity BOM balloon with a quantity greater than 1.
3. Type the quantity that you want to subtract from this balloon and use to create another balloon for the same component.
4. Select an attachment point for the new balloon by selecting an occurrence of the component that does not have a balloon.
    Note: You cannot select for attachment any silhouette edges of torii that are not perpendicular to the screen.
5. Select a location on the sheet to place the new balloon. The system creates the new balloon, displaying the quantity that you subtracted from the first balloon.

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